Simulation Models and Analyses Reference

Summary

This comprehensive reference describes the simulation models and types of analyses available using Altium Designer's Mixed-Signal Circuit Simulator.

Technical Reference
TR0113 (v1.6) April 21, 2008

This reference details the simulation models and circuit simulation analyses and describes some simulation troubleshooting techniques.

Simulation Models

The Altium Designer-based Circuit Simulator is a true mixed-signal simulator, meaning that it can analyze circuits that include both analog and digital devices.

The Simulator uses an enhanced version of the event-driven XSpice, developed by the Georgia Tech Research Institute (GTRI), which itself is based on Berkeley's SPICE3 code. It is fully SPICE3f5 compatible, as well as providing support for a range of PSpice® device models.

Model Types

The models supported by the Simulator can be effectively grouped into the following categories:

SPICE3f5 analog models

These are predefined analog device models that are built-in to SPICE. They cover the various common analog component types, such as resistors, capacitors and inductors, as well as voltage and current sources, transmission lines and switches. The five most common semiconductor devices are also modeled - diodes, BJTs, JFETs, MESFETs and MOSFETs.

A large number of model files (`*.mdl`) are also included, that define the behavior of specific instances of these devices.

PSpice analog models

These are predefined analog device models that are built-in to PSpice. To support these models, changes have been made to the general form for the corresponding SPICE3f5 device and/or additional parameter support has been added for use in a linked model file.

Note: These models are not listed separately in this reference. PSpice support information is included as part of the information for the relevant SPICE3f5 device model.

XSpice analog models

These are predefined analog device code models that are built-in to XSpice. Code models allow the specification of complex, non-ideal device characteristics, without the need to develop long-winded sub-circuit definitions that can adversely affect Simulator speed performance. The supplied models cover special functions such as gain, hysteresis, voltage and current limiting and definitions of s-domain transfer functions.

The SPICE prefix for these models is A.

Sub-Circuit models

These are models for more complex devices, such as operational amplifiers, timers, crystals, etc, that have been described using the hierarchical sub-circuit syntax.

A sub-circuit consists of SPICE elements that are defined and referenced in a fashion similar to device models. There is no limit on the size or complexity of sub-circuits and sub-circuits can call other sub-circuits. Each sub-circuit is defined in a sub-circuit file (`*.ckt`).

The SPICE prefix for theses models is X.

Digital models

These are digital device models that have been created using the Digital SimCode™ language. This is a special descriptive language that allows digital devices to be simulated using an extended version of the event-driven XSpice. It is a form of the standard XSpice code model.

Source SimCode model definitions are stored in an ASCII text file (`*.txt`). Compiled SimCode models are stored in a compiled model file (`*.scb`). Multiple device models can be placed in the same file, with each reference by means of a special "func=" parameter.

The SPICE prefix for theses models is A.

Digital SimCode is a proprietary language - devices created with it are not compatible with other simulators, nor are digital components created for other simulators compatible with the Altium Designer-based mixed-signal Simulator.

Notes

For more detailed information concerning SPICE, PSpice and XSpice, consult the respective user manuals for each. The XSpice manual is particularly useful for learning about the syntax required for the Code Models added to XSpice by GTRI and extensions that have been made to SPICE3.

Many of the component libraries (`*.IntLib`) that come with the installation, feature simulation-ready devices. These devices have the necessary model or sub-circuit file included and linked to the schematic component. These are pure SPICE models for maximum compatibility with analog simulators.

There were no syntax changes made between SPICE3f3 and SPICE3f5. The manual for SPICE3f3 therefore describes the correct syntax for the netlist and models supported by the Altium Designer-based mixed-signal Simulator.

Component and Simulation Multipliers

When entering a value for a component or other simulation-related parameter, the value can be entered in one of the following formats:

- As an integer value (e.g. 10)
- As a floating point value (e.g. 3.142)
- As an integer or floating point value followed by an integer exponent (e.g. 10E-2, 3.14E2)
- As an integer or floating point value followed by a valid scale factor

With respect to the last format, the following is a list of valid scale factors (multipliers) that can be used:

Scale Factor	Represents
T	10^{12}
G	10^{9}
Meg	10^{6}
K	10^{3}
mil	25.4^{-6}
m	10^{-3}
u (or µ)	10^{-6}
n	10^{-9}
p	10^{-12}
f	10^{-15}

Notes

Letters immediately following a value that are not valid scale factors will be ignored.

Letters immediately following a valid scale factor are also ignored. They can be beneficial as a reference to measurement units used, when viewing the component on the schematic and the relevant parameter is made visible.

The scale factor must immediately follow the value - spaces are not permitted.

The scale factors may be entered in either lower or upper case, or a mixture thereof.

Examples

10, 10V 10Volts and 10Hz all represent the same number, 10. The letters are ignored in all cases as none of them are valid scale factors.

M, m, MA, MSec and MMhos all represent the same scale factor, 10-3. In each case, the letters after the first "m" are ignored.

1000, 1000.0, 1000Hz, 1e3, 1.0e3, 1KHz and 1K all represent the same number, 1000.

Simulation-ready Components - Quick Reference

Within the vast array of integrated libraries supplied as part of the Altium Designer installation, a great number of schematic components are simulation-ready. This means they have a linked simulation model and are ready (with default parameters) to be placed on a schematic sheet, with a view to circuit simulation using the Altium Designer-based Mixed-Signal Simulator.

Simulation-ready schematic components fall into two categories - those supplied specifically for simulation or as part of a generic default set of such components and those that are part of integrated libraries supplied by a specific manufacturer.

The following sections provide a full listing of the non-manufacturer-specific, simulation-ready schematic components that are supplied as part of the installation.

Simulation Sources

The following schematic components can be found in the Simulation Sources integrated library (`Library\Simulation\Simulation Sources.IntLib`).

Component	Description	Model Name	Model File	SPICE Prefix
.IC	Initial Condition	ControlStatement	Not Required	None
.NS	Node Set	ControlStatement	Not Required	None
BISRC	Non-Linear Dependent Current Source	NLDS	Not Required	B
BVSRC	Non-Linear Dependent Voltage Source	NLDS	Not Required	B
DSEQ	Data Sequencer with clock output	xsourcesub	XSourceSub.ckt	X
DSEQ2	Data Sequencer	xsourcesub2	XSourceSub2.ckt	X
ESRC	Voltage Controlled Voltage Source	VCVS	Not Required	E
FSRC	Current Controlled Current Source	CCCS	Not Required	F
GSRC	Voltage Controlled Current Source	VCCS	Not Required	G
HSRC	Current Controlled Voltage Source	CCVS	Not Required	H
IEXP	Exponential Current Source	IEXP	Not Required	I
IPULSE	Pulse Current Source	IPULSE	Not Required	I
IPWL	Piecewise Linear Current Source	IPWL	Not Required	I
ISFFM	Frequency Modulated Sinusoidal Current Source	ISFFM	Not Required	I
ISIN	Sinusoidal Current Source	ISIN	Not Required	I
ISRC	DC Current Source	ISRC	Not Required	I
VEXP	Exponential Voltage Source	VEXP	Not Required	V
VPULSE	Pulse Voltage Source	VPULSE	Not Required	V
VPWL	Piecewise Linear Voltage Source	VPWL	Not Required	V
VSFFM	Frequency Modulated Sinusoidal	VSFFM	Not Required	V

Component	Description	Model Name	Model File	SPICE Prefix
	Voltage Source			
VSIN	Sinusoidal Voltage Source	VSIN	Not Required	V
VSRC	DC Voltage Source	VSRC	Not Required	V
VSRC2	DC Voltage Source with pin 2 connected to Ground by default and the following parameter defaults: Value = 5V AC Magnitude = 1V AC Phase = 0	VSRC	Not Required	V

Simulation Transmission Lines

The following schematic components can be found in the Simulation Transmission Line integrated library (`\Library\Simulation\Simulation Transmission Line.IntLib`).

Component	Description	Model Name	Model File	SPICE Prefix
LLTRA	Lossless transmission line	LLTRA	Not Required	T
LTRA	Lossy transmission line	LTRA	LTRA.mdl	O
URC	Uniform distributed lossy line	URC	URC.mdl	U

Simulation Math Functions

The following schematic components can be found in the Simulation Math Function integrated library (`\Library\Simulation\Simulation Math Function.IntLib`).

Component	Description	Model Name	Model File	SPICE Prefix
ABSI	Absolute value of current	ABSI	ABSI.ckt	X
ABSV	Absolute value of voltage (single-ended input)	ABSV	ABSV.ckt	X
ABSVR	Absolute value of voltage (differential input)	ABSVR	ABSVR.ckt	X
ACOSHI	Hyperbolic arc cosine of current	ACOSHI	ACOSHI.ckt	X
ACOSHV	Hyperbolic arc cosine of voltage (single-ended input)	ACOSHV	ACOSHV.ckt	X
ACOSHVR	Hyperbolic arc cosine of voltage (differential input)	ACOSHVR	ACOSHVR.ckt	X
ACOSI	Arc cosine of current	ACOSI	ACOSI.ckt	X
ACOSV	Arc cosine of voltage (single-ended input)	ACOSV	ACOSV.ckt	X
ACOSVR	Arc cosine of voltage (differential input)	ACOSVR	ACOSVR.ckt	X
ADDI	Addition of currents	ADDI	ADDI.ckt	X
ADDV	Addition of voltages (single-ended inputs)	ADDV	ADDV.ckt	X
ADDVR	Addition of voltages (differential inputs)	ADDVR	ADDVR.ckt	X
ASINHI	Hyperbolic arc sine of current	ASINHI	ASINHI.ckt	X
ASINHV	Hyperbolic arc sine of voltage (single-ended	ASINHV	ASINHV.ckt	X

Component	Description	Model Name	Model File	SPICE Prefix
	input)			
ASINHVR	Hyperbolic arc sine of voltage (differential input)	ASINHVR	ASINHVR.ckt	X
ASINI	Arc sine of current	ASINI	ASINI.ckt	X
ASINV	Arc sine of voltage (single-ended input)	ASINV	ASINV.ckt	X
ASINVR	Arc sine of voltage (differential input)	ASINVR	ASINVR.ckt	X
ATANHI	Hyperbolic arc tangent of current	ATANHI	ATANHI.ckt	X
ATANHV	Hyperbolic arc tangent of voltage (single-ended input)	ATANHV	ATANHV.ckt	X
ATANHVR	Hyperbolic arc tangent of voltage (differential input)	ATANHVR	ATANHVR.ckt	X
ATANI	Arc tangent of current	ATANI	ATANI.ckt	X
ATANV	Arc tangent of voltage (single-ended input)	ATANV	ATANV.ckt	X
ATANVR	Arc tangent of voltage (differential input)	ATANVR	ATANVR.ckt	X
COSHI	Hyperbolic cosine of current	COSHI	COSHI.ckt	X
COSHV	Hyperbolic cosine of voltage (single-ended input)	COSHV	COSHV.ckt	X
COSHVR	Hyperbolic cosine of voltage (differential input)	COSHVR	COSHVR.ckt	X
COSI	Cosine of current	COSI	COSI.ckt	X
COSV	Cosine of voltage (single-ended input)	COSV	COSV.ckt	X
COSVR	Cosine of voltage (differential input)	COSVR	COSVR.ckt	X
DIVI	Division of currents	DIVI	DIVI.ckt	X
DIVV	Division of voltages (single-ended inputs)	DIVV	DIVV.ckt	X
DIVVR	Division of voltages (differential inputs)	DIVVR	DIVVR.ckt	X
EXPI	Exponential of current	EXPI	EXPI.ckt	X
EXPV	Exponential of voltage (single-ended input)	EXPV	EXPV.ckt	X
EXPVR	Exponential of voltage (differential input)	EXPVR	EXPVR.ckt	X
LNI	Natural logarithm of current	LNI	LNI.ckt	X
LNV	Natural logarithm of voltage (single-ended input)	LNV	LNV.ckt	X
LNVR	Natural logarithm of voltage (differential input)	LNVR	LNVR.ckt	X
LOGI	Logarithm of current	LOGI	LOGI.ckt	X
LOGV	Logarithm of voltage (single-ended input)	LOGV	LOGV.ckt	X
LOGVR	Logarithm of voltage (differential input)	LOGVR	LOGVR.ckt	X
MULTI	Multiplication of currents	MULTI	MULTI.ckt	X

Component	Description	Model Name	Model File	SPICE Prefix
MULTV	Multiplication of voltages (single-ended input)	MULTV	MULTV.ckt	X
MULTVR	Multiplication of voltages (differential input)	MULTVR	MULTVR.ckt	X
SINHI	Hyperbolic sine of current	SINHI	SINHI.ckt	X
SINHV	Hyperbolic sine of voltage (single-ended input)	SINHV	SINHV.ckt	X
SINHVR	Hyperbolic sine of voltage (differential input)	SINHVR	SINHVR.ckt	X
SINI	Sine of current	SINI	SINI.ckt	X
SINV	Sine of voltage (single-ended input)	SINV	SINV.ckt	X
SINVR	Sine of voltage (differential input)	SINVR	SINVR.ckt	X
SQRTI	Square root of current	SQRTI	SQRTI.ckt	X
SQRTV	Square root of voltage (single-ended input)	SQRTV	SQRTV.ckt	X
SQRTVR	Square root of voltage (differential input)	SQRTVR	SQRTVR.ckt	X
SUBI	Subtraction of currents	SUBI	SUBI.ckt	X
SUBV	Subtraction of voltages (single-ended inputs)	SUBV	SUBV.ckt	X
SUBVR	Subtraction of voltages (differential inputs)	SUBVR	SUBVR.ckt	X
TANI	Tangent of current	TANI	TANI.ckt	X
TANV	Tangent of voltage (single-ended input)	TANV	TANV.ckt	X
TANVR	Tangent of voltage (differential input)	TANVR	TANVR.ckt	X
UNARYI	Unary minus of current	UNARYI	UNARYI.ckt	X
UNARYV	Unary minus of voltage (single-ended input)	UNARYV	UNARYV.ckt	X
UNARYVR	Unary minus of voltage (differential input)	UNARYVR	UNARYVR.ckt	X

Simulation Special Functions

The following schematic components can be found in the Simulation Special Function integrated library (`\Library\Simulation\Simulation Special Function.IntLib`).

Component	Description	Model Name	Model File	SPICE Prefix
CLIMITER	Controlled Limiter (single-ended current or voltage I/O)	CLIMIT	Not Required	A
CLIMITERR	Controlled Limiter (differential current or voltage I/O)	CLIMIT	Not Required	A
CMETER	Capacitance meter (single-ended current or voltage I/O)	CMETER	Not Required	A
CMETERR	Capacitance meter (differential current or voltage I/O)	CMETER	Not Required	A
DDT	Differentiator block (single-ended current or voltage I/O)	D_DT	Not Required	A
DDTR	Differentiator block (differential current or	D_DT	Not Required	A

Component	Description	Model Name	Model File	SPICE Prefix
	voltage I/O)			
DIVIDE	Two-quadrant divider (single-ended current or voltage I/O)	DIVIDE	Not Required	A
DIVIDER	Two-quadrant divider (differential current or voltage I/O)	DIVIDE	Not Required	A
FTOV	Frequency to Voltage converter	FTOV	FTOV.ckt	X
GAIN	Simple gain block with optional offsets (single-ended current or voltage I/O)	GAIN	Not Required	A
GAINR	Simple gain block with optional offsets (differential current or voltage I/O)	GAIN	Not Required	A
HYSTERESIS	Hysteresis block (single-ended current or voltage I/O)	HYST	Not Required	A
HYSTERESISR	Hysteresis block (differential current or voltage I/O)	HYST	Not Required	A
ILIMIT	Current limiter (single-ended voltage input, single-ended conductance output)	ILIMIT	Not Required	A
ILIMITR	Current limiter (differential voltage input, differential conductance output)	ILIMIT	Not Required	A
INT	Integrator block (single-ended current or voltage I/O)	INT	Not Required	A
INTR	Integrator block (differential current or voltage I/O)	INT	Not Required	A
ISW	Current controlled switch	ISW	ISW.mdl	W
LIMITER	Limiter block (single-ended current or voltage I/O)	LIMIT	Not Required	A
LIMITERR	Limiter block (differential current or voltage I/O)	LIMIT	Not Required	A
LMETER	Inductance meter (single-ended current or voltage I/O)	LMETER	Not Required	A
LMETERR	Inductance meter (differential current or voltage I/O)	LMETER	Not Required	A
MULT	Multiplier block (single-ended current or voltage I/O)	MULT	MULT.ckt	X
MULTR	Multiplier block (differential current or voltage I/O)	MULTR	MULTR.ckt	X
ONESHOT	Controlled oneshot (single-ended current or voltage I/O)	ONESHOT	Not Required	A
ONESHOTR	Controlled oneshot (differential current or voltage I/O)	ONESHOT	Not Required	A
PWL	Piece-wise linear controlled source (single-ended current or voltage I/O)	PWL	Not Required	A
PWLR	Piece-wise linear controlled source (differential	PWL	Not Required	A

Component	Description	Model Name	Model File	SPICE Prefix
	current or voltage I/O)			
SLEWRATE	Simple slew-rate block (single-ended current or voltage I/O)	SLEW	Not Required	A
SLEWRATER	Simple slew-rate block (differential current or voltage I/O)	SLEW	Not Required	A
SUM	Summer block (single-ended current or voltage I/O)	SUM	SUM.ckt	X
SUMR	Summer block (differential current or voltage I/O)	SUMR	SUMR.ckt	X
SXFER	S-domain transfer function (single-ended current or voltage I/O)	S_XFER	Not Required	A
SXFERR	S-domain transfer function (differential current or voltage I/O)	S_XFER	Not Required	A
VCO-Sine	Sinusoidal voltage controlled oscillator	SINEVCO	SINEVCO.ckt	X
VCO-Sqr	Square voltage controlled oscillator	SQRVCO	SQRVCO.ckt	X
VCO-Tri	Triangular voltage controlled oscillator	TRIVCO	TRIVCO.ckt	X
VSW	Voltage controlled switch	VSW	VSW.mdl	S

Miscellaneous Devices

The following schematic components can be found in the Miscellaneous Devices integrated library (`\Library\Miscellaneous Devices.IntLib`).

Component	Description	Model Name	Model File	SPICE Prefix
2N3904	NPN General Purpose Amplifier	2N3904	2N3904.mdl	Q
2N3906	PNP General Purpose Amplifier	2N3906	2N3906.mdl	Q
ADC-8	Generic 8-bit A/D Converter	ADC8	ADC8.mdl	A
Bridge1	Diode Bridge	BRIDGE	Bridge.ckt	X
Bridge2	Full Wave Diode Bridge	BRIDGE	Bridge.ckt	X
Cap	Capacitor	CAP	Not Required	C
Cap2	Capacitor	CAP	Not Required	C
Cap Pol1	Polarized Capacitor (Radial)	CAP	Not Required	C
Cap Pol2	Polarized Capacitor (Axial)	CAP	Not Required	C
Cap Pol3	Polarized Capacitor (Surface Mount)	CAP	Not Required	C
Cap Semi	Semiconductor Capacitor with default value = 100pF	CAP	CAP.mdl	C
D Schottky	Schottky Diode	SKYDIODE	SKYDIODE.mdl	D
D Varactor	Variable Capacitance Diode	BBY31	BBY31.mdl	D

Component	Description	Model Name	Model File	SPICE Prefix
D Zener	Zener Diode	ZENER	ZENER.mdl	D
DAC-8	Generic 8-bit D/A Converter	DAC8	DAC8.mdl	A
Diode	Default Diode	DIODE	DIODE.mdl	D
Diode 1N914	High Conductance Fast Diode	1N914	1N914.mdl	D
Diode 1N4001	1 Amp General Purpose Rectifier	1N4001	1N4001.mdl	D
Diode 1N4001	1 Amp General Purpose Rectifier	1N4002	1N4002.mdl	D
Diode 1N4001	1 Amp General Purpose Rectifier	1N4003	1N4003.mdl	D
Diode 1N4001	1 Amp General Purpose Rectifier	1N4004	1N4004.mdl	D
Diode 1N4001	1 Amp General Purpose Rectifier	1N4005	1N4005.mdl	D
Diode 1N4001	1 Amp General Purpose Rectifier	1N4006	1N4006.mdl	D
Diode 1N4001	1 Amp General Purpose Rectifier	1N4007	1N4007.mdl	D
Diode 1N4148	High Conductance Fast Diode	1N4148	1N4148.mdl	D
Diode 1N4149	Computer Diode	1N4149	1N4149.mdl	D
Diode 1N4150	High Conductance Ultra Fast Diode	1N4150	1N4150.mdl	D
Diode 1N4448	High Conductance Fast Diode	1N4448	1N4448.mdl	D
Diode 1N4934	1 Amp Fast Recovery Rectifier	1N4934	1N4934.mdl	D
Diode 1N5400	3 Amp General Purpose Rectifier	1N5400	1N5400.mdl	D
Diode 1N5401	3 Amp General Purpose Rectifier	1N5401	1N5401.mdl	D
Diode 1N5402	3 Amp General Purpose Rectifier	1N5402	1N5402.mdl	D
Diode 1N5404	3 Amp General Purpose Rectifier	1N5404	1N5404.mdl	D
Diode 1N5406	3 Amp General Purpose Rectifier	1N5406	1N5406.mdl	D
Diode 1N5407	3 Amp Medium Power Silicon Rectifier Diode	1N5407	1N5407.mdl	D
Diode	3 Amp Medium Power Silicon Rectifier	1N5408	1N5408.mdl	D

Component	Description	Model Name	Model File	SPICE Prefix
1N5408	Diode			
Diode 10TQ035	Schottky Rectifier	10TQ035	10TQ035.mdl	D
Diode 10TQ040	Schottky Rectifier	10TQ040	10TQ040.mdl	D
Diode 10TQ045	Schottky Rectifier	10TQ045	10TQ045.mdl	D
Diode 11DQ03	Schottky Rectifier	11DQ03	11DQ03.mdl	D
Diode 18TQ045	Schottky Rectifier	18TQ045	18TQ045.mdl	D
Diode BAS16	Silicon Switching Diode for High-Speed Switching	BAS16	BAS16.ckt	X
Diode BAS21	Silicon Switching Diode for High-Speed, High-Voltage Switching	BAS21	BAS21.mdl	D
Diode BAS70	Silicon AF Schottky Diode for High-Speed Switching	BAS70	BAS70.mdl	D
Diode BAS116	Silicon Low Leakage Diode	BAS116	BAS116.mdl	D
Diode BAT17	Silicon RF Schottky Diode for Mixer Applications in the VHF/UHF Range	BAT17	BAT17.mdl	D
Diode BAT18	Low-Loss RF Switching Diode	BAT18	BAT18.mdl	D
Diode BBY31	SOT23 Silicon Planar Variable Capacitance Diode	BBY31	BBY31.mdl	D
Diode BBY40	SOT23 Silicon Planar Variable Capacitance Diode	BBY40	BBY40.mdl	D
Dpy 16-Seg	13.7mm Gray Surface As AlInGaP Red Alphanumeric Display: 2-Character, CC	HDSP_A27C	HDSP_A27C.ckt	X
D Tunnel1	Tunnel Diode - RLC Model	DTUNNEL1	DTUNNEL1.ckt	X
D Tunnel2	Tunnel Diode - Dependent Source Model	DTUNNEL2	DTUNNEL2.ckt	X
Dpy Amber-CA	7.62mm Black Surface Orange 7-Segment Display: CA, RH DP	HDSP_A211	HDSP_A211.ckt	X
Dpy Amber-CC	7.62mm Black Surface Orange 7-Segment Display: CC, RH DP	HDSP_A213	HDSP_A213.ckt	X
Dpy Blue-CA	14.2mm General Purpose Blue 7-Segment Display: CA, RH DP, Gray Surface	HDSP_501B	HDSP_501B.ckt	X
Dpy Blue-CC	14.2mm General Purpose Blue 7-Segment Display: CC, RH DP, Gray Surface	HDSP_503B	HDSP_503B.ckt	X
Dpy Green-CA	7.62mm Black Surface Green 7-Segment Display: CA, RH DP	HDSP_A511	HDSP_A511.ckt	X
Dpy Green-	7.62mm Black Surface Green 7-Segment	HDSP_A513	HDSP_A513.ckt	X

Component	Description	Model Name	Model File	SPICE Prefix
CC	Display: CC, RH DP			
Dpy Overflow	7.62mm HER 7-Segment Display: Universal +/-1 Overflow, RH DP	5082_7616	5082_7616.ckt	X
Dpy Red-CA	7.62mm Black Surface HER 7-Segment Display: CA, RH DP	HDSP_A211	HDSP_A211.ckt	X
Dpy Red-CC	7.62mm Black Surface HER 7-Segment Display: CC, RH DP	HDSP_A213	HDSP_A213.ckt	X
Dpy Yellow-CA	7.6mm Micro Bright Yellow 7-Segment Display: CA, RH DP	HDSP_7401	HDSP_7401.ckt	X
Dpy Yellow-CC	7.6mm Micro Bright Yellow 7-Segment Display: CC, RH DP	HDSP_7403	HDSP_7403.ckt	X
Fuse 1	Fuse	FUSE	FUSE.ckt	X
Fuse 2	Fuse	FUSE	FUSE.ckt	X
IGBT-N	Insulated Gate Bipolar Junction Transistor (N-Channel)	IRGPC40U	IRGPC40U.ckt	X
IGBT-P	Insulated Gate Bipolar Junction Transistor (P-Channel)	PIGBT	PIGBT.ckt	X
Inductor	Inductor	INDUCTOR	Not Required	L
Inductor Adj	Adjustable Inductor	INDUCTOR	Not Required	L
Inductor Iron	Magnetic-core Inductor	INDUCTOR	Not Required	L
Inductor Iron Adj	Adjustable Magnetic-core Inductor	INDUCTOR	Not Required	L
JFET-N	N-Channel JFET	NJFET	NJFET.mdl	J
JFET-P	P-Channel JFET	PJFET	PJFET.mdl	J
LED0	Typical INFRARED GaAs LED	LED0	LED0.mdl	D
LED1	Typical RED GaAs LED	LED1	LED1.mdl	D
LED2	Typical RED, GREEN, YELLOW, AMBER GaAs LED	LED2	LED2.mdl	D
LED3	Typical BLUE SiC LED	LED3	LED3.mdl	D
MESFET-N	N-Channel MESFET	NMESFET	NMESFET.mdl	Z
MESFET-P	P-Channel MESFET	PMESFET	PMESFET.mdl	Z
MOSFET -N	N-Channel MOSFET	NMOS	NMOS.mdl	M
MOSFET -N4	N-Channel MOSFET (externally terminated substrate)	NMOS	NMOS.mdl	M
MOSFET -P	P-Channel MOSFET	PMOS	PMOS.mdl	M
MOSFET -P4	P-Channel MOSFET (externally terminated substrate)	PMOS	PMOS.mdl	M
NMOS-2	N-Channel Power MOSFET	IRF1010	IRF1010.ckt	X

Component	Description	Model Name	Model File	SPICE Prefix
NPN	NPN Bipolar Junction Transistor	NPN	NPN.mdl	Q
NPN1	NPN Darlington Bipolar Junction Transistor	NPN1	NPN1.ckt	X
NPN2	NPN Darlington Bipolar Junction Transistor	NPN2	NPN2.ckt	X
NPN3	NPN Darlington Bipolar Junction Transistor	NPN3	NPN3.ckt	X
Op Amp	FET Operational Amplifier	AD645A	AD645A.ckt	X
Optoisolator2	Optoisolator	OPTOISO	OPTOISO.ckt	X
PLL	Generic Phase Locked Loop	PLLx	PLLx.ckt	X
PMOS-2	P-Channel Power MOSFET	IRF9510	IRF9510.ckt	X
PNP	PNP Bipolar Junction Transistor	PNP	PNP.mdl	Q
PNP1	PNP Darlington Bipolar Junction Transistor	PNP1	PNP1.ckt	X
PNP2	PNP Darlington Bipolar Junction Transistor	PNP2	PNP2.ckt	X
PNP3	PNP Darlington Bipolar Junction Transistor	PNP3	PNP3.ckt	X
PUT	Programmable Unijunction Transistor	PUT	PUT.ckt	X
QNPN	NPN Bipolar Junction Transistor	QNPN	QNPN.mdl	Q
Relay	Single-Pole Double-Throw Relay	SPDTRELAY	SPDTRELAY.ckt	X
Relay-DPDT	Double Pole Double Throw Relay	DPDTRELAY	DPDTRELAY.ckt	X
Relay-DPST	Double-Pole Single Throw Relay	DPSTRELAY	DPSTRELAY.ckt	X
Relay-SPDT	Single-Pole Double-Throw Relay	SPDTRELAY	SPDTRELAY.ckt	X
Relay-SPST	Single-Pole Single-Throw Relay	SPSTRELAY	SPSTRELAY.ckt	X
Res1	Resistor	RESISTOR	Not Required	R
Res2	Resistor	RESISTOR	Not Required	R
Res Adj1	Variable Resistor	VRES	Not Required	R
Res Adj2	Variable Resistor	VRES	Not Required	R
Res Bridge	Resistor Bridge		Not Required	R
Res Pack1	Resistor Array (parts)	RESISTOR	Not Required	R
Res Pack2	Resistor Array (parts)	RESISTOR	Not Required	R
Res Pack3	Resistor Array	respack_8	respack_8.ckt	X
Res Pack4	Resistor Array	respack_8	respack_8.ckt	X
Res Semi	Semiconductor Resistor	RES	RES.mdl	R
Res Tap	Tapped Resistor	POT	Not Required	R
RPot	Potentiometer	POT	Not Required	R
SCR	Silicon Controlled Rectifier	SCR	SCR.ckt	X
SW-DIP4	4-way DIP Switch (thru-hole)	dpsw4	DIPSW4.ckt	X
SW-DIP8	8-way DIP Switch (thru-hole)	dpsw8	DIPSW8.ckt	X

Component	Description	Model Name	Model File	SPICE Prefix
SW DIP-2	2-way DIP Switch (thru-hole)	dpsw2	DIPSW2.ckt	X
SW DIP-3	3-way DIP Switch (thru-hole)	dpsw3	DIPSW3.ckt	X
SW DIP-4	4-way DIP Switch (surface mount)	dpsw4	DIPSW4.ckt	X
SW DIP-5	5-way DIP Switch (thru-hole)	dpsw5	DIPSW5.ckt	X
SW DIP-6	6-way DIP Switch (thru-hole)	dpsw6	DIPSW6.ckt	X
SW DIP-7	7-way DIP Switch (thru-hole)	dpsw7	DIPSW7.ckt	X
SW DIP-8	8-way DIP Switch (surface mount)	dpsw8	DIPSW8.ckt	X
SW DIP-9	9-way DIP Switch (thru-hole)	dpsw9	DIPSW9.ckt	X
Trans	Transformer	TRANSFORMER	Not Required	K
Trans Adj	Variable Transformer	TRANSFORMER	Not Required	K
Trans BB	Buck-boost Transformer (Ideal)	IDEAL4W	IDEAL4W.ckt	X
Trans CT	Center-Tapped Transformer (Coupled Inductor Model)	TRANSFORMER	Not Required	K
Trans CT Ideal	Center-Tapped Transformer (Ideal)	IDEALTRANSCT	IDEALTRANSCT.ckt	X
Trans Cupl	Transformer (Coupled Inductor Model)	TRANSFORMER	Not Required	K
Trans Eq	Transformer (Equivalent Circuit Model)	TRANS	TRANS.ckt	X
Trans Ideal	Transformer (Ideal)	IDEALTRANS	IDEALTRANS.ckt	X
Trans3	Three-winding transformer (non-ideal)	NI3WTRANS	NI3WTRANS.ckt	X
Trans3 Ideal	Three-winding transformer (ideal)	IDEAL3W	IDEAL3W.ckt	X
Trans4	Four-winding transformer (non-ideal)	NI4WTRANS	NI4WTRANS.ckt	X
Trans4 Ideal	Four-winding transformer (ideal)	IDEAL4W	IDEAL4W.ckt	X
Triac	Silicon Bidirectional Triode Thyristor	MAC15A8	MAC15A8.ckt	X
Tube 6L6GC	Beam Power Pentode	6L6GC	6L6GC.ckt	X
Tube 6SN7	Medium Mu Dual Triode	6SN7	6SN7.ckt	X
Tube 12AU7	Medium Mu Dual Triode	12AU7	12AU7.ckt	X
Tube 12AX7	High Mu Dual Triode	12AX7	12AX7.ckt	X
Tube 7199	Medium Mu Triode and Sharp Cutoff Pentode	7199	7199.ckt	X
UJT-N	Unijunction transistor with N-type base	NUJT	NUJT.ckt	X
XTAL	Crystal Oscillator	XTAL	XTAL.ckt	X

Simulation-Ready Components by Manufacturer

The following sections provide a listing of various manufacturer-specific integrated libraries that are supplied as part of the installation and which contain simulation-ready schematic components. The sub-folders containing the integrated libraries (arranged by Manufacturer) can be found along the following path:

`\Library\` - on the drive to which you installed the software.

Note that not all schematic components in a listed library may have a linked simulation model.

Agilent Technologies

Agilent LED Display 7-Segment, 1-Digit.IntLib
Agilent LED Display 7-Segment, 2-Digit.IntLib
Agilent LED Display 7-Segment, 3-Digit.IntLib
Agilent LED Display 7-Segment, 4-Digit.IntLib
Agilent LED Display Alphanumeric.IntLib
Agilent LED Display Digit & Word Icon.IntLib
Agilent LED Display Overflow.IntLib
Agilent Optoelectronic LED.IntLib

Analog Devices

- AD Amplifier Buffer.IntLib
- AD Analog Multiplier Divider.IntLib
- AD Audio Pre-Amplifier.IntLib
- AD Differential Amplifier.IntLib
- AD Instrumentation Amplifier.IntLib
- AD Operational Amplifier.IntLib
- AD Power Mgt Voltage Reference.IntLib
- AD RF and IF Modulator Demodulator.IntLib
- AD Variable Gain Amplifier.IntLib
- AD Video Amplifier.IntLib

Burr-Brown

- BB Amplifier Buffer.IntLib
- BB Analog Integrator.IntLib
- BB Differential Amplifier.IntLib
- BB Instrumentation Amplifier.IntLib
- BB Isolation Amplifier.IntLib
- BB Logarithmic Amplifier.IntLib
- BB Operational Amplifier.IntLib
- BB Transconductance Amplifier.IntLib
- BB Universal Active Filter.IntLib
- BB Voltage Controlled Amplifier.IntLib

ECS

- ECS Crystal Oscillator.IntLib

Elantec

- Elantec Amplifier Buffer.IntLib
- Elantec Analog Comparator.IntLib
- Elantec Analog Multiplier Divider.IntLib
- Elantec Interface Line Transceiver.IntLib
- Elantec Operational Amplifier.IntLib
- Elantec Video Amplifier.IntLib
- Elantec Video Gain Control Circuit.IntLib

Fairchild Semiconductor

- FSC Discrete BJT.IntLib
- FSC Discrete Diode.IntLib
- FSC Discrete Rectifier.IntLib
- FSC Interface Display Driver.IntLib
- FSC Interface Line Transceiver.IntLib
- FSC Logic Buffer Line Driver.IntLib
- FSC Logic Counter.IntLib
- FSC Logic Decoder Demux.IntLib
- FSC Logic Flip-Flop.IntLib
- FSC Logic Gate.IntLib
- FSC Logic Latch.IntLib
- FSC Logic Multiplexer.IntLib
- FSC Logic Parity Gen Check Detect.IntLib
- FSC Logic Register.IntLib

Infineon

- Infineon Discrete BJT.IntLib
- Infineon Discrete Diode.IntLib

International Rectifier

- IR Discrete IGBT.IntLib
- IR Discrete MOSFET - Half Bridge.IntLib
- IR Discrete MOSFET - Low Power.IntLib
- IR Discrete MOSFET - Power.IntLib
- IR Discrete SCR.IntLib
- IR Rectifier - Schottky.IntLib
- IR Rectifier - Standard Recovery.IntLib
- IR Rectifier - Ultrafast Recovery.IntLib

Intersil

- Intersil Discrete BJT.IntLib
- Intersil Discrete MOSFET.IntLib
- Intersil Operational Amplifier.IntLib

KEMET Electronics

- KEMET Chip Capacitor.IntLib

Linear Technology

- LT Amplifier Buffer.IntLib
- LT Operational Amplifier.IntLib
- LT Video Amplifier.IntLib

Maxim

- Maxim Amplifier Buffer.IntLib
- Maxim Analog Comparator.IntLib
- Maxim Communication Receiver.IntLib
- Maxim Current-Feedback Amplifier.IntLib
- Maxim Multiplexed Video Amplifier.IntLib
- Maxim Operational Amplifier.IntLib

- Maxim Video Amplifier.IntLib
- Maxim Wideband Amplifier.IntLib

Motorola

- Motorola Amplifier Operational Amplifier.IntLib
- Motorola Discrete BJT.IntLib
- Motorola Discrete Diode.IntLib
- Motorola Discrete IGBT.IntLib
- Motorola Discrete JFET.IntLib
- Motorola Discrete MOSFET.IntLib
- Motorola Discrete SCR.IntLib
- Motorola Discrete TRIAC.IntLib

National Semiconductor

- NSC Amplifier Buffer.IntLib
- NSC Analog Comparator.IntLib
- NSC Converter Analog to Digital.IntLib
- NSC Discrete BJT.IntLib
- NSC Discrete Diode.IntLib
- NSC Discrete JFET.IntLib
- NSC Discrete Rectifier.IntLib
- NSC Interface Display Driver.IntLib
- NSC Interface Line Transceiver.IntLib
- NSC Logic Arithmetic.IntLib
- NSC Logic Buffer Line Driver.IntLib
- NSC Logic Comparator.IntLib
- NSC Logic Counter.IntLib
- NSC Logic Decoder Demux.IntLib
- NSC Logic Flip-Flop.IntLib
- NSC Logic Gate.IntLib
- NSC Logic Latch.IntLib
- NSC Logic Multiplexer.IntLib
- NSC Logic Parity Gen Check Detect.IntLib
- NSC Logic Register.IntLib
- NSC Operational Amplifier.IntLib
- NSC Power Mgt Voltage Regulator.IntLib

Panasonic

- Panasonic Resistor.IntLib

Philips

- Philips Discrete BJT - Darlington.IntLib
- Philips Discrete BJT - Lower Power.IntLib
- Philips Discrete BJT - Medium Power.IntLib
- Philips Discrete BJT - RF Transistor.IntLib
- Philips Discrete Diode - Schottky.IntLib
- Philips Discrete Diode - Switching.IntLib
- Philips Discrete JFET.IntLib
- Philips Discrete MOSFET - Lower Power.IntLib

- Philips Discrete MOSFET - Power.IntLib

Raltron Electronics

- Raltron Crystal Oscillator.IntLib

ST Microelectronics

- ST Discrete BJT.IntLib
- ST Interface Display Driver.IntLib
- ST Logic Arithmetic.IntLib
- ST Logic Buffer Line Driver.IntLib
- ST Logic Comparator.IntLib
- ST Logic Counter.IntLib
- ST Logic Decoder.IntLib
- ST Logic Flip-Flop.IntLib
- ST Logic Gate.IntLib
- ST Logic Latch.IntLib
- ST Logic Multiplexer.IntLib
- ST Logic Register.IntLib
- ST Operational Amplifier.IntLib

Teccor Electronics

- Teccor Discrete SCR.IntLib
- Teccor Discrete TRIAC.IntLib

Texas Instruments

- TI Analog Comparator.IntLib
- TI Interface 8-bit Line Transceiver.IntLib
- TI Interface Display Driver.IntLib
- TI Interface Line Transceiver.IntLib
- TI Logic Arithmetic.IntLib
- TI Logic Buffer Line Driver.IntLib
- TI Logic Comparator.IntLib
- TI Logic Counter.IntLib
- TI Logic Decoder Demux.IntLib
- TI Logic Flip-Flop.IntLib
- TI Logic Gate 1.IntLib
- TI Logic Gate 2.IntLib
- TI Logic Latch.IntLib
- TI Logic Multiplexer.IntLib
- TI Logic Parity Gen Check Detect.IntLib
- TI Logic Register.IntLib
- TI Operational Amplifier.IntLib
- TI Power Mgt Voltage Regulator.IntLib

Toshiba

- Toshiba Discrete IGBT.IntLib

Vishay

- Vishay Cera-Mite Ceramic Axial-Lead Capacitor.IntLib
- Vishay Draloric Ceramic Disc Capacitor.IntLib

- Vishay Draloric Ceramic Tubular Capacitor.IntLib
- Vishay Electrolytic Radial-Lead Capacitor.IntLib
- Vishay Lite-On Discrete Diode.IntLib
- Vishay Roederstein Electrolytic Axial-Lead Capacitor.IntLib
- Vishay Roederstein Electrolytic Radial-Lead Capacitor.IntLib
- Vishay Roederstein Electrolytic Snap-In Pins Capacitor.IntLib
- Vishay Roederstein Electrolytic Solder Ring Capacitor.IntLib
- Vishay Roederstein Tantalum Radial-Lead Capacitor.IntLib
- Vishay Siliconix Discrete JFET.IntLib
- Vishay Siliconix Discrete MOSFET.IntLib
- Vishay Sprague Tantalum Axial-Lead Capacitor.IntLib
- Vishay Sprague Tantalum Chip Capacitor.IntLib
- Vishay Sprague Tantalum Radial-Lead Capacitor.IntLib
- Vishay Tansitor Tantalum Axial-Lead Capacitor.IntLib
- Vishay Tansitor Tantalum Radial-Lead Capacitor.IntLib
- Vishay Telefunken Discrete Diode.IntLib
- Vishay Vitramon Ceramic Dipped Capacitor.IntLib

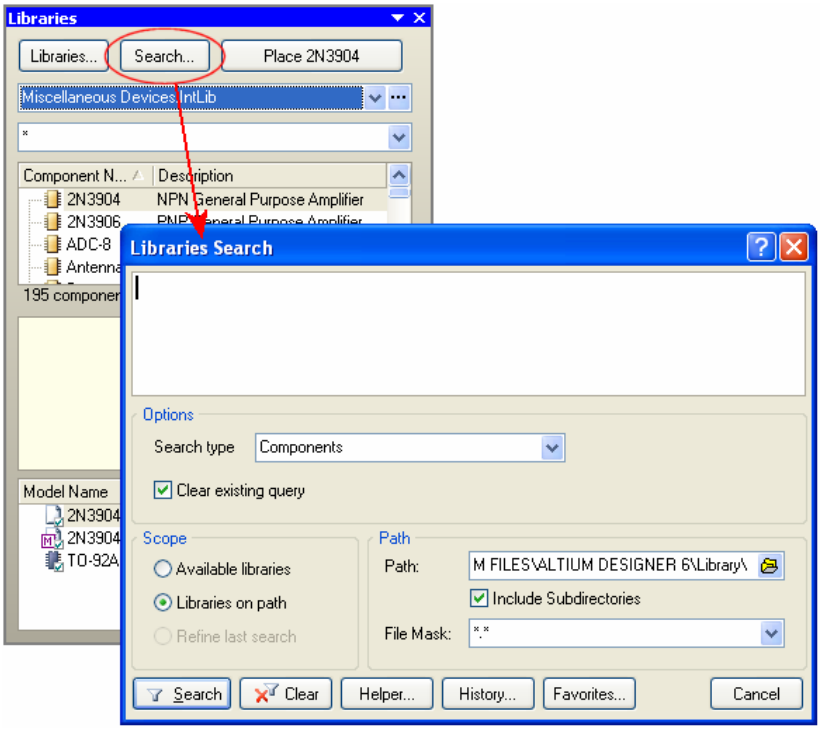
Zetex

- Zetex Discrete BJT.IntLib
- Zetex Discrete Diode.IntLib
- Zetex Discrete MOSFET.IntLib

Searching for Simulation-Ready Components

With such a vast collection of simulation-ready components – both generic and manufacturer-specific – scattered across a multitude of integrated libraries, you may think that finding the component you need is like searching for that proverbial 'needle in a haystack'. To simplify this process, search features are available that allow you to:

- Search for a simulation-ready component across libraries local to your installation of Altium Designer
- Search for a simulation-ready component within the available up-to-date libraries supplied by the Altium Library Development Center (ALDC) – available from the Altium Website.



Searching Installation Libraries

By using Altium Designer's Libraries Search feature, you can quickly search for simulation-ready components:

- across all available libraries for the active project
- in libraries along a specified search path.

Search criteria is specified within the *Libraries Search* dialog. Access this dialog from within a schematic document (**Tools » Find Component**) or by clicking the **Search** button on the **Libraries** panel.

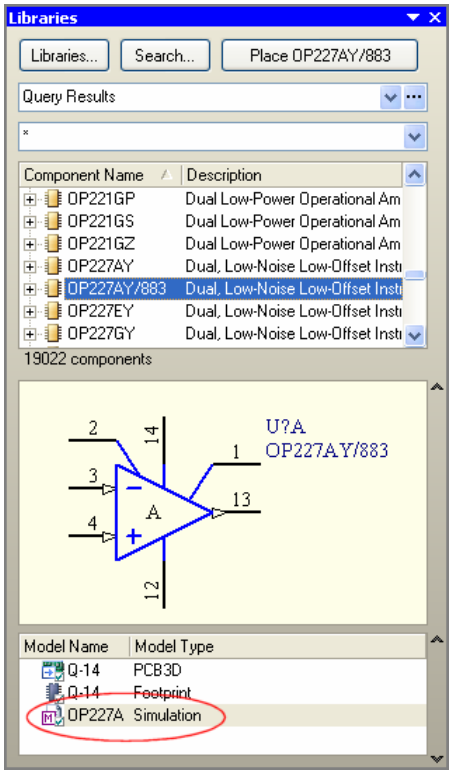
By defining suitable queries, you can quickly find the simulation-ready components that you require. The more specific the query, the narrower the search and the greater the probability of returning favorable results.

The coarsest search you could do is to search for all components that have a linked simulation model. To do this, you would simply enter the following query into the top section of the *Libraries Search* dialog (often referred to as the Query Editor section):

```
HasModel('SIM','*','FALSE')
```

This expression returns all components that have a linked simulation model. The `'*'` entry is used as a wildcard for the model name. The `'FALSE'` entry specifies that the model need not be the current model for the component (if multiple simulation models have been defined and linked to the component).

If you are unfamiliar with Altium Designer's Query language, use the Query Helper to help you construct the required query expression (accessed by clicking the **Helper** button).



The results of the search will be listed in the **Libraries** panel, under a new entry to the libraries drop-down list – `Query Results`.

The library search facility also offers the ability to refine the last search made. This enables you to apply further, perhaps more specific search criteria, to the list of Query Results obtained by the previous search. For example, you might set up a search targeting the entire `\Library` directory for all components that have a linked simulation model and a description containing the word `Diode`, using the following query expression:

```
HasModel('SIM','*','FALSE') AND (Description Like '*Diode*')
```

Such a search might well yield in excess of 1500 components fitting that criteria! You might then decide that out of these returned results, you wish to further search and return only Zener diodes. Simply enable the **Refine last search** option in the Scope region of the *Libraries Search* dialog, enter the following new query expression and click **Search**.

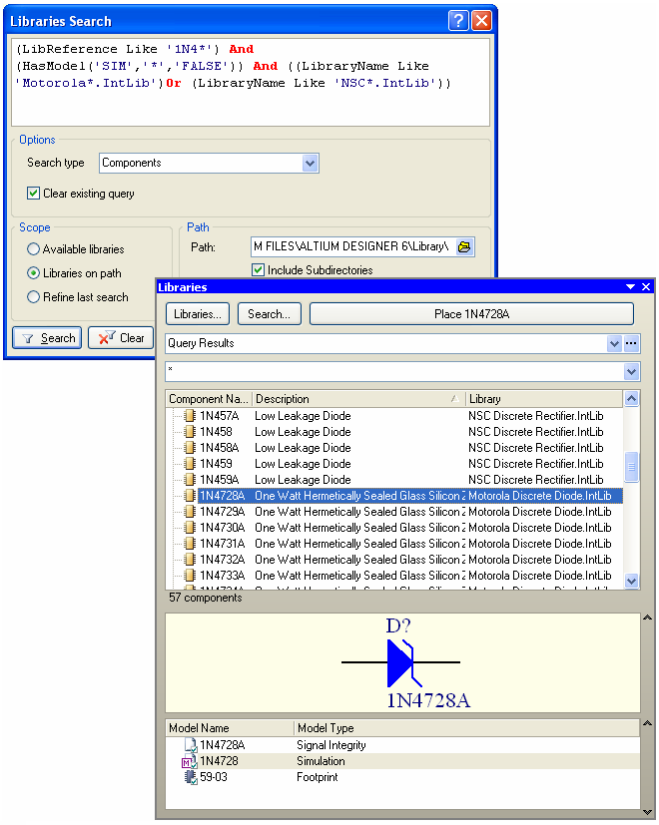
```
Description Like '*Zener*'
```

The subset of the previous query results falling under the scope of the new, refining query, will be displayed as the new Query Results list in the **Libraries** panel.

Let us now consider a more complex search. You might know a partial name for the component you need for a design and have preferred manufacturers that you like to use. Again, searching is simplified through the power of the Query language. Consider a design where you need to use a particular diode, whose name is of the form `1N4*`. You need to search for all components with a name based on this root, which are simulation ready, and which are manufactured by either National Semiconductor or Motorola. This search can be specified by entering the following query:

```
(LibReference Like '1N4*') And (HasModel('SIM','*','FALSE')) And ((LibraryName Like
'NSC*.IntLib') Or (LibraryName Like 'Motorola*.IntLib'))
```

Out of a possible 18000+ simulation-ready components that come installed with Altium Designer, the preceding query returns a manageable 57 components, as illustrated by the following image.



For help on getting started with writing query expressions, refer to the *Introduction to the Query Language* article.

For more detailed information regarding queries, refer to the article *An Insider's Guide to the Query Language*.

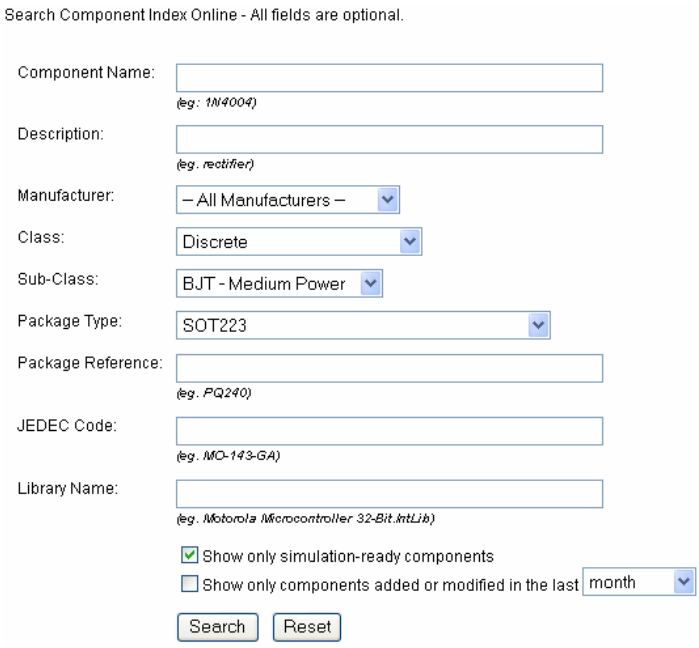
For detailed information on query language syntax, including example query expressions for each keyword, refer to the *Query Language Reference*.

For detailed information about the Libraries panel, refer to the *Libraries panel* section of the *Altium Designer Panels Reference*.

Searching via the Altium Website

By navigating to the *Altium Designer Libraries* area of the Altium Website, you can browse, search and download up-to-date Altium Designer integrated libraries for board-level design. Simply click on the available link for the **Altium Designer board-level design integrated libraries** and then access the **Search for a component** facility. Use this facility to quickly search for simulation-ready components.

Use the fields provided to make your search criteria as broad or specific as required. If, for example, you wanted to quickly find all simulation-ready components of a particular type – across all manufacturer integrated libraries – simply set the **Class** and **Sub-Class** fields as required and ensure that the Show only simulation-ready components option is enabled. Such a search could be narrowed further by entering a specific package type for the component, and so on.



After running the search, the results will be listed, alphabetically, and by manufacturer. The following information is provided:

- Component Name
- Manufacturer of the component
- Description of the component
- Package Type
- Name of the integrated library into which the component has been compiled
- Downloadable zip file – containing all integrated libraries for that manufacturer
- Date Component was last updated.

Integrated Library Component Search Result(s):

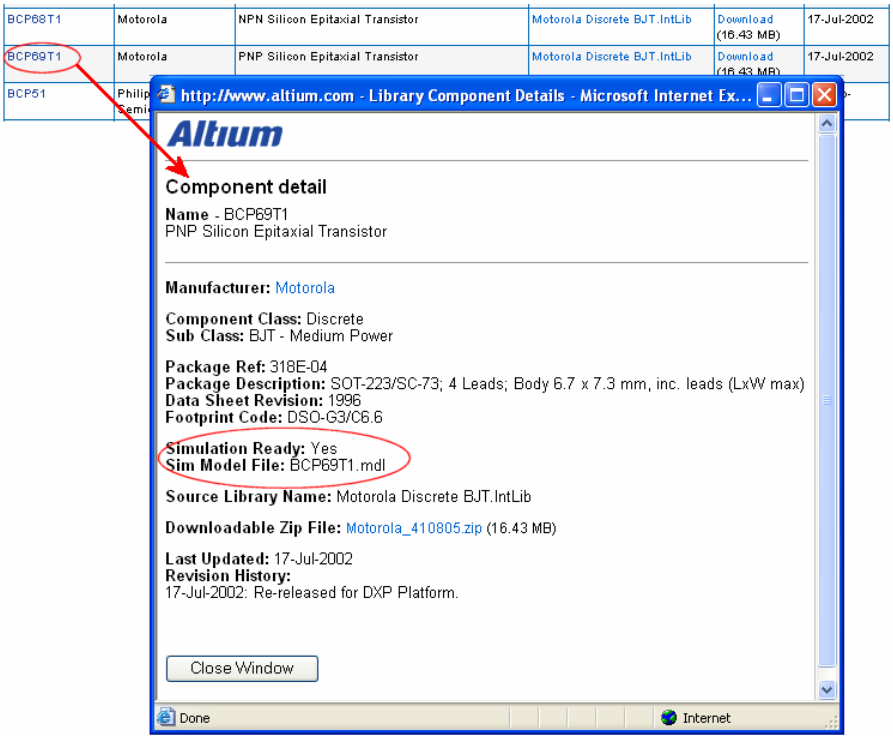
50 matches found

Class: **Discrete**
Sub-Class: **BJT - Medium Power**
Package Type: **SOT223**

Simulation-ready components only.

Component Name Click for info	Manufacturer	Description	Library Name Click for contents	Zip File	Last Updated
BCP52	Fairchild Semiconductor	PNP General Purpose Amplifier	FSC Discrete BJT.IntLib	Download (3.62 MB)	17-Jul-2002
BCP54	Fairchild Semiconductor	NPN General Purpose Amplifier	FSC Discrete BJT.IntLib	Download (3.62 MB)	17-Jul-2002
NZT651	Fairchild Semiconductor	NPN Current Driver Transistor	FSC Discrete BJT.IntLib	Download (3.62 MB)	17-Jul-2002
NZT751	Fairchild Semiconductor	PNP Current Driver Transistor	FSC Discrete BJT.IntLib	Download (3.62 MB)	17-Jul-2002
BF720	Infineon Technologies	NPN Silicon High-Voltage Transistor	Infineon Discrete BJT.IntLib	Download (103 KB)	8-Jun-2000
BF721	Infineon Technologies	PNP Silicon High-Voltage Transistor	Infineon Discrete BJT.IntLib	Download (103 KB)	8-Jun-2000
BCP53T1	Motorola	PNP Silicon Epitaxial Transistor	Motorola Discrete BJT.IntLib	Download (16.43 MB)	17-Jul-2002
BCP56T1	Motorola	NPN Silicon Epitaxial Transistor	Motorola Discrete BJT.IntLib	Download (16.43 MB)	17-Jul-2002
BCP68T1	Motorola	NPN Silicon Epitaxial Transistor	Motorola Discrete BJT.IntLib	Download (16.43 MB)	17-Jul-2002
BCP69T1	Motorola	PNP Silicon Epitaxial Transistor	Motorola Discrete BJT.IntLib	Download (16.43 MB)	17-Jul-2002
BCP51	Philips Semiconductors	PNP Medium Power Transistor	Philips Discrete BJT - Medium Power.IntLib	Download (792 KB)	17-Sep-2003
BCP54-16	Philips Semiconductors	NPN Medium Power Transistor	Philips Discrete BJT - Medium Power.IntLib	Download (792 KB)	17-Sep-2003
BCP55-16	Philips Semiconductors	NPN Medium Power Transistor	Philips Discrete BJT - Medium Power.IntLib	Download (792 KB)	17-Sep-2003
BCP56-16	Philips Semiconductors	NPN Medium Power Transistor	Philips Discrete BJT - Medium Power.IntLib	Download (792 KB)	17-Sep-2003
BCP68	Philips Semiconductors	NPN Medium Power Transistor	Philips Discrete BJT - Medium Power.IntLib	Download (792 KB)	17-Sep-2003
BCP69	Philips Semiconductors	PNP Medium Power Transistor	Philips Discrete BJT - Medium Power.IntLib	Download (792 KB)	17-Sep-2003
BDL31	Philips Semiconductors	NPN BISS-Transistor	Philips Discrete BJT - Medium Power.IntLib	Download (792 KB)	17-Sep-2003
BDL32	Philips Semiconductors	PNP BISS-Transistor	Philips Discrete BJT - Medium Power.IntLib	Download (792 KB)	17-Sep-2003
BDP31	Philips Semiconductors	NPN Medium Power Transistor	Philips Discrete BJT - Medium Power.IntLib	Download (792 KB)	17-Sep-2003
BDP32	Philips Semiconductors	PNP Medium Power Transistor	Philips Discrete BJT - Medium Power.IntLib	Download (792 KB)	17-Sep-2003

From the list of results, you can access further information about a component, simply by clicking on the entry for its name. From a simulation perspective, this gives you information about any model or sub-circuit file linked as a model to the component.



The Netlist Template - Explained

The Netlist Template allows access to the information that is entered into the XSpice netlist for a given component. It is accessed by clicking on the **Netlist Template** tab, at the bottom of the *Sim Model* dialog.

For all of the predefined model kinds and sub-kinds, the Netlist Template is read-only. If, however, one of these predefined entries does not allow enough control over the information placed in the netlist, you can define your own template.

To edit the Netlist Template, you need to select `Generic Editor` in the **Model Sub-Kind** region of the *Sim Model* dialog - ensuring that the **Model Kind** field is first set to `General`. This will be the default model kind/sub-kind setting when adding a new simulation model to a schematic component. For all other General model sub-kinds, you can effectively change to Generic Editor and edit the predefined template - massaging it to your own requirements.

When defining the Netlist Template, the information entered should be in accordance with the requirements of SPICE3f5/XSpice and the syntax rules described below.

Netlist Template Syntax

Characters that are entered into the template are written to the XSpice netlist verbatim, except for the following special characters:

% - percent sign

@ - commercial at

& - ampersand

? - question mark

~ - tilde

- number sign

These characters are translated when creating the netlist, as shown in the following table:

Syntax in Netlist Template...	Netlister replaces with...
`@<param>`	Value of `<param>`. An error is raised if a parameter of this name does not exist or if there is no value assigned to it.
`&<param>`	Value of `<param>`. No error is raised if the parameter is undefined.

Syntax in Netlist Template...	Netlister replaces with...
?<param>s…s	Text between s…s separators if <param> is defined.
?<param>s…ss…s	Text between first s…s separators if <param> is defined, else the second s…s separators.
~<param>s…s	Text between s…s separators if <param> is NOT defined.
~<param>s…ss…s	Text between first s…s separators if <param> is NOT defined, else the second s…s separators.
#<param>s…s	Text between s…s separators if <param> is defined, but ignore the rest of the template if <param> is NOT defined.
#s…s	Text between s…s separators if there is any text to be entered into the XSpice netlist from subsequent entries in the Netlist Template.
%<pin id>	The net name of the net to which the schematic pin mapped to <pin id> connects.
%%	A literal percent character.

In the above table,

- s represents a separator character (, . ; / |).
- <param> refers to the name of a parameter.

If the parameter name contains any non-alphanumeric characters, it should be enclosed in double quotes. For example:

@"DC Magnitude" - double quotes used here because the name contains a space.

&"Init_Cond" - double quotes used here because the name contains an underscore.

Double quotes should also be used when you wish to add an alphanumeric prefix to a parameter name. For example:

@"DESIGNATOR"A - the use of the double quotes ensures that A is appended to the component designator.

Syntax Examples

The following are examples of the special character syntax entries in the previous table. Information is given in each case, about how the syntax entry is translated by the Netlister.

@"AC Phase"

The parameter name AC Phase is enclosed in braces because of the space. This will be replaced in the netlist with the value of the AC Phase parameter. If there is no parameter of this name, or its value is blank then an error will be given.

&Area

If a parameter named Area exists and has a value, then it's value will be entered into the netlist. If the parameter is undefined (i.e. either it does not exist or has no value assigned) then nothing will be written to the netlist, but no error will be raised. This can be used for optional parameters.

?IC|IC=@IC|

If the parameter named IC is defined then the text within the || separators will be inserted into the netlist. For example if the parameter IC had value 0.5 then IC=0.5 would be inserted into the netlist in place of this entry. If the parameter is undefined then nothing will be inserted into the netlist.

?IC/IC=@IC//IC=0/

This is the same as the previous example, except that if the parameter IC is undefined then IC=0 will be inserted into the netlist. Note also that a different separator character has been used.

~VALUE/1k/

If a parameter named VALUE is NOT defined then the text 1k will be inserted into the netlist.

~VALUE/1k//@VALUE/

This is the same as the previous example, except that if the parameter VALUE is defined then its text value will be inserted into the netlist.

#"AC Magnitude"|AC@"AC Magnitude"|@"AC Phase"

This example can be seen in the predefined netlist template for the sinusoidal voltage source.

If the AC Magnitude parameter has been defined then the contents of the separators is evaluated and inserted into the netlist. All following entries in the netlist are also evaluated and entered into the netlist (in this case @"AC Phase").

If for example AC Magnitude=1 and AC Phase=0 then AC 1 0 will be inserted into the netlist. If, however, AC Phase was undefined, an error would be raised.

If the parameter AC Magnitude is undefined then nothing following the #"AC Magnitude" entry in the netlist template will be entered into the netlist.

#|PARAMS:|?Resistance|Resistance=Resistance|?Current|Current=@Current|

This example can be seen in the predefined netlist template for a parameterized subcircuit (see F1 in `Fuse.PrjPcb`).

If the Resistance and Current parameters are both undefined then there will be no text to be inserted into the netlist following the #|PARAMS:| entry, so the text in the separators will be omitted also.

If for example the parameters have values Resistance=1k and Current=5mA then this will result in text following the #|PARAMS:| entry and `PARAMS: Resistance=1k Current=5mA` will be the entry made in the netlist.

@DESIGNATOR%1%2@VALUE

This example is to demonstrate the use of the % character.

If for example the parameters have values DESIGNATOR=R1 and VALUE=1k, and the pins are mapped on the **Port Map** tab of the *Sim Model* dialog according to the following table:

Schematic Pin	Model Pin	Net name to which Schematic Pin connects
1 (N+)	1 (1)	GND
2 (N-)	2 (2)	OUT

Then the text `R1 GND OUT 1k` will be placed into the XSpice netlist for this component.

Checking the Netlist Template

To check the Netlist Template, simply click on the **Netlist Preview** tab at the bottom of the *Sim Model* dialog. The text displayed in this tab is exactly as it will be written to the XSpice netlist file when a netlist is generated or a simulation is run. The following exception applies:

- If you are in the Schematic Library Editor, or the document/project has not been compiled, the net names that the model pins map to will not be available. In this case, the schematic pin designators are inserted, enclosed in <> braces.

Any errors that occur while parsing user-defined entries in the Netlist Template will also be displayed, so that any errors can be resolved prior to exiting the dialog.

PSpice Support

To facilitate compatibility with PSpice, support for various additional PSpice-based functions and operators is provided, as well as the use of global parameters – to represent values in a PSpice-modeled circuit.

Additional Function Support

The following additional functions are supported:

ARCTAN(x)	-	returns the inverse tangent of x
ATAN2(y, x)	-	returns the inverse tangent of y/x
IF(t, x, y)	-	If t is TRUE then x, ELSE y
LIMIT(x, min, max)	-	while min < x < max, x is returned If x < min, min is returned If x > max, max is returned
LOG10(x)	-	returns the decimal logarithm of x
MAX(x, y)	-	returns the maximum of x and y
MIN(x, y)	-	returns the minimum of x and y

PWR(x, y) - returns x to the power of y

PWRS(x, y) - returns signed x to the power of y:

If x > 0, the result is positive

If x < 0, the result is negative.

SCHEDULE(x_1, y_1,…x_n, y_n) - allows you to control the value of y based on time x. An entry for time = 0s must be entered.

From time = x_1 to x_2, returns y_1

From time = x_2 to x_3, returns y_2, and so on.

SGN(x) - returns the sign of x (a.k.a. the signum function).

If x < 0, returns -1

If x = 0, returns 0

If x > 0, returns 1

STP(x) - unit step function.

If x > 0, returns 1

If x < 0, returns 0

TABLE(x, x_1, y_1,…x_n, y_n) - allows you to construct a look-up table, returning the y value corresponding to x when all x_n, y_n points are plotted and connected by straight lines.

If x > than the largest x value in the table, then the y value associated to that x value will be returned.

If x < than the smallest x value in the table, then the y value associated to that x value will be returned.

Additional Operator Support

The following additional operators are supported:

- ** (exponentiation)
- == (equality test)
- != (non-equality test)
- & (Boolean AND)
- | (Boolean OR)

.PARAM Support

The PSpice .PARAM statement is supported. This statement defines the value of a parameter, allowing you to use a parameter name in place of numeric values for a circuit description. Parameters can be constants, expressions or a combination of the two. A single parameter statement can include reference to one or more additional parameter statements.

In addition, the following three internal variables (predefined parameters) are available for use in expressions:

GMIN - shunt conductance for semiconductor p-n junctions.

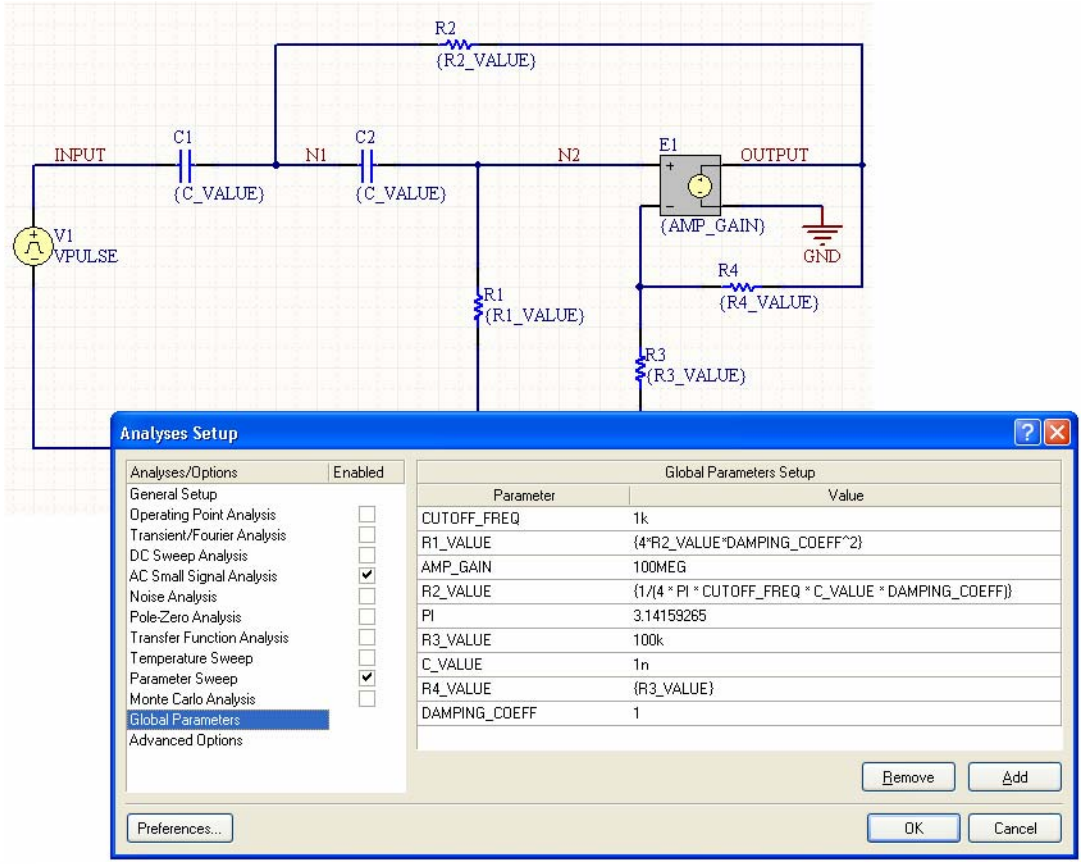
TEMP - temperature.

VT - thermal voltage.

Global Parameters

Altium Designer's Circuit Simulator supports the use of global parameters and equations. Use a global parameter in an equation and then use that equation in a component value on your schematic. Alternatively, define the equation as a global parameter and then reference the global parameter from a component value.

Simply include the expression or parameter name within curly braces {} – when the Simulator detects this it will attempt to evaluate it, checking the Global Parameters page of the Simulator's *Analyses Setup* dialog for the definition of any part of the expression that cannot be immediately resolved.



For an example of using global parameters and equations in a simulation, refer to the example project `Global Params.PrjPCB`, which can be found in the `\Examples\Circuit Simulation\PSpice Examples\Global Parameters` folder of the installation.

SPICE3f5 models

These are predefined analog device models that are built-in to SPICE. They cover the following, common analog component types.

General

- *Capacitor*
- *Capacitor (Semiconductor)*
- *Coupled Inductors*
- *Diode*
- *Inductor*
- *Potentiometer*
- *Resistor*
- *Resistor (Semiconductor)*
- *Resistor (Variable)*

Transistors

- *Bipolar Junction Transistor (BJT)*
- *Junction Field-Effect Transistor (JFET)*
- *Metal Semiconductor Field-Effect Transistor (MESFET)*
- *Metal Oxide Semiconductor Field-Effect Transistor (MOSFET)*

Switches

- *Current Controlled Switch*
- *Voltage Controlled Switch*

Transmission Lines

- *Lossless Transmission Line*
- *Lossy Transmission Line*
- *Uniform Distributed RC (lossy) Transmission Line*

Current Sources

- *Current-Controlled Current Source*
- *DC Current Source*
- *Exponential Current Source*
- *Frequency Modulated Sinusoidal Current Source*
- *Non-Linear Dependent Current Source*
- *Piecewise Linear Current Source*
- *Pulse Current Source*
- *Sinusoidal Current Source*
- *Voltage-Controlled Current Source*

Voltage Sources

- *Current-Controlled Voltage Source*
- *DC Voltage Source*
- *Exponential Voltage Source*
- *Frequency Modulated Sinusoidal Voltage Source*
- *Non-Linear Dependent Voltage Source*
- *Piecewise Linear Voltage Source*
- *Pulse Voltage Source*
- *Sinusoidal Voltage Source*

- *Voltage-Controlled Voltage Source*

Initial Conditions

- *Initial Condition*
- *Nodeset*

Notes

Many of the models have associated model files (`*.mdl`). A model file is used to allow specification of specific device parameters (e.g. on and off resistances for a switch).

Many of the above models have been modified to make them compatible with PSpice. In such cases, PSpice support information is included as part of that model's information, later in this reference.

Many of the component libraries (`*.IntLib`) that come with the installation, feature simulation-ready devices. These devices have the necessary model or sub-circuit file included and linked to the schematic component. These are pure SPICE models for maximum compatibility with analog simulators.

For more detailed information regarding SPICE3, consult the SPICE3f5 User Manual. There were no syntax changes made between SPICE3f3 and SPICE3f5. The manual for SPICE3f3 therefore describes the correct syntax for the netlist and models supported by the Altium Designer-based Mixed-Signal Simulator.

General

Capacitor

Model Kind

General

Model Sub-Kind

Capacitor

SPICE Prefix

C

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @VALUE ?"INITIAL VOLTAGE"|IC=@"INITIAL VOLTAGE"|
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

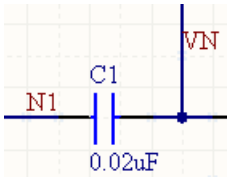
Value - value for the capacitance (in Farads).

Initial Voltage - time-zero voltage of capacitor (in Volts).

Notes

The value for the `Initial Voltage` only applies if the **Use Initial Conditions** option is enabled on the **Transient/Fourier Analysis Setup** page of the *Analyses Setup* dialog.

Examples



Consider the capacitor in the above image, with the following characteristics:

- Pin1 (positive) is connected to net `N1`
- Pin2 (negative) is connected to net `VN`
- Designator is `C1`

- Value = `0.02uF`.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
C1 N1 VN 0.02uF
```

Capacitor (Semiconductor)

Model Kind

General

Model Sub-Kind

Capacitor(Semiconductor)

SPICE Prefix

C

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 &VALUE &MODEL ?LENGTH|L=@LENGTH| ?WIDTH|W=@WIDTH| ?"INITIAL
VOLTAGE"|IC=@"INITIAL VOLTAGE"|
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Value - value for the capacitance (in Farads).

Length - length of the capacitor (in meters).

Width - width of the capacitor (in meters) (Default = 1e-6).

Initial Voltage - time-zero voltage of capacitor (in Volts).

Parameters (definable within model file)

The following is a list of process-related parameters that can be stored in the associated model file:

CJ - junction bottom capacitance (in F/meters^2).

CJSW - junction sidewall capacitance (in F/meters).

DEFW - default width (in meters). - this value will be overridden by a value entered for **Width** in the *Sim Model* dialog.

NARROW - narrowing due to side etching (in meters). (Default = 0).

Notes

The value for the `Initial Voltage` only applies if the **Use Initial Conditions** option is enabled on the **Transient/Fourier Analysis Setup** page of the *Analyses Setup* dialog.

You can specify either a direct value for the capacitance OR enter values for the capacitors' length and width. In the case of the latter, a value for the capacitance will be calculated, in conjunction with parameter information stored in the model.

The equation used to calculate the capacitance from geometric data is:

```
CAP = CJ(LENGTH - NARROW)(WIDTH - NARROW) + 2CJSW(LENGTH + WIDTH - 2 NARROW)
```

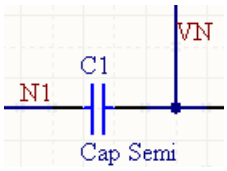
If a direct value for capacitance is not specified, the model name and length must be supplied in order for the geometric-based capacitance value to be calculated.

The link to the required model file (`*.mdl`) is specified on the **Model Kind** tab of the *Sim Model* dialog. The Model Name is used in the netlist to reference this file.

Either the direct capacitance value OR the geometric data used to calculate it can be entered, but not both.

Where a parameter has an indicated default (as part of the SPICE model definition), that default will be used if no value is specifically entered. The default should be applicable to most simulations. Generally you do not need to change this value.

Examples



Consider the semiconductor capacitor in the above image, with the following characteristics:

- Pin1 is connected to net `N1`
- Pin2 is connected to net `VN`
- Designator is `C1`
- The linked simulation model file is `CAP.mdl`

If a value for the capacitance was entered directly, say 100 pF, and no other parameters were specified on the **Parameters** tab of the *Sim Model* dialog, then the entry in the SPICE netlist would be:

```
*Schematic Netlist:

C1 N1 VN 100pF CAP
```

Consider now, instead of entering a direct value for the capacitance, the following parameters were defined in the *Sim Model* dialog:

- **Length** = `10u`
- **Width** = `1u`

the entry in the netlist would be:

```
C1 N1 VN CAP L=10u W=1u
```

The value for the capacitance will be calculated accurately using the geometric data specified and any further parameter definitions in the model file (`CAP.mdl`).

PSpice Support

To make this device model compatible with PSpice, the following additional model parameters are supported and can be entered into a linked model file (`*.mdl`) for the device:

C - capacitance multiplier. (Default = 1).

TC1 - linear temperature coefficient (in $^{\circ}C^{-1}$). (Default = 0).

TC2 - quadratic temperature coefficient (in $^{\circ}C^{-2}$). (Default = 0).

VC1 - linear voltage coefficient (in $Volt^{-1}$). (Default = 0).

VC2 - quadratic voltage coefficient (in $Volt^{-2}$). (Default = 0).

Where a parameter has an indicated default, that default will be used if no value is specifically entered.

The following parameters – common to most devices in PSpice – are not supported:

T_ABS

T_MEASURED

T_REL_GLOBAL

T_REL_LOCAL.

The format for the PSpice model file is:

`.MODEL` *ModelName* `CAP(`*Model Parameters*`)`,

where

- *ModelName* is the name of the model, the link to which is specified on the **Model Kind** tab of the *Sim Model* dialog. This name is used in the netlist (`&MODEL`) to reference the required model in the linked model file.
- *Model Parameters* are a list of supported parameters for the model, entered with values as required.

For an example of using a PSpice-compatible capacitor model in a simulation, refer to the example project `Capacitor.PrjPCB`, which can be found in the `\Examples\Circuit Simulation\PSpice Examples\Capacitor` folder of the installation.

Coupled Inductors

Model Kind

General

Model Sub-Kind

Coupled Inductors

SPICE Prefix

K

SPICE Netlist Template Format

```
LA_@DESIGNATOR %1 %2 @"INDUCTANCE A"
LB_@DESIGNATOR %3 %4 @"INDUCTANCE B"
@DESIGNATOR LA_@DESIGNATOR LB_@DESIGNATOR @"COUPLING FACTOR"
```

Parameters (definable at component level)

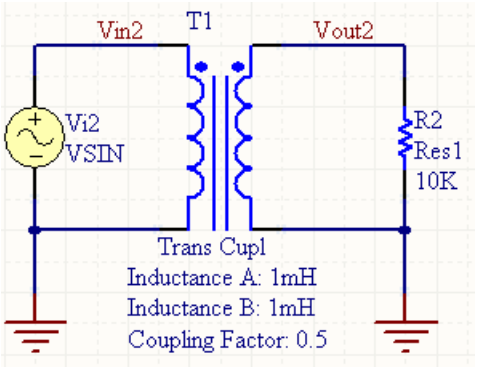
The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Inductance A - value for the inductance of discrete inductor A (in Henrys).

Inductance B - value for the inductance of discrete inductor B (in Henrys).

Coupling Factor - the coupling coefficient, representing the flux linkage between the windings of the two individual inductors. Permissible values lie in the range 0<CF≤1, where 1 (the ideal) means all flux linking inductor A also links inductor B.

Examples



Consider the transformer in the above image, which uses a coupled inductor model and has the following characteristics:

- The positive pin of the Primary is connected to net `Vin2`
- The negative pin of the Primary is connected to net `GND`
- The positive pin of the secondary is connected to net `Vout2`
- The negative pin of the secondary is connected to net `GND`
- Designator is `T1`
- Inductance A = `1mH`
- Inductance B = `1mH`
- Coupling Factor = `0.5`.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
LA_KT1 VIN2 0 1mH
LB_KT1 VOUT2 0 1mH
KT1 LA_KT1 LB_KT1 0.5
```

Placing a transformer with a coupled inductor model is probably the simplest and quickest way of adding a coupled inductor to your source schematic. However, should you wish to place individual inductors in the schematic design and couple them, the required information for the SPICE netlist can be readily supplied through the use of a Text Frame.

Simply place a text frame in the schematic document and ensure that the first line is:

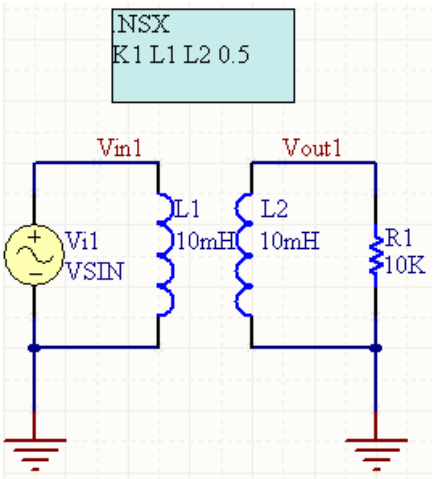
```
.NSX
```

Then type the information required - each line will be included in the netlist, exactly as it is written, appearing between the following comment entries:

```
*Begin NSX text frames
*End of NSX text frames
```

The following example illustrates the use of two inductors and a text frame to provide a coupled inductor for simulation purposes.

Consider the circuit in the adjacent image, which contains two discrete inductors with the following characteristics:



- Designator (Primary inductor) = `L1`
- Designator (Secondary inductor) = `L2`
- The positive pin of L1 is connected to net `Vin1`
- The negative pin of L1 is connected to net `GND`
- The positive pin of L2 is connected to net `Vout1`
- The negative pin of L2 is connected to net `GND`
- Value for inductance (L1) = `10mH`
- Value for inductance (L2) = `10mH`

The entries in the text frame are dissected as follows:

- `.NSX` - the mandatory first line that tells the netlister that the following lines are extra information to be added when generating the SPICE netlist.
- `K1` - the designator for the 'coupled inductor', where K is the required SPICE prefix.
- `L1, L2` - the designators of the two individual inductors
- `0.5` - the value for the Coupling Factor

The entry in the SPICE netlist would be:

```
*Begin NSX text frames
K1 L1 L2 0.5
*End of NSX text frames
.
.
*Schematic Netlist:
L1 VIN1 0 10mH
L2 VOUT1 0 10mH
```

Diode

Model Kind

General

Model Sub-Kind

Diode

SPICE Prefix

D

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @MODEL &"AREA FACTOR" &"STARTING CONDITION" ?"INITIAL VOLTAGE"|IC=@"INITIAL VOLTAGE"| ?TEMPERATURE|TEMP=@TEMPERATURE|
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Area Factor - specifies the number of equivalent parallel devices of the specified model. This setting affects a number of parameters in the model.

Starting Condition - set to `OFF` to set diode voltage to zero during operating point analysis. Can be useful as an aid in convergence.

Initial Voltage - time-zero voltage across the diode (in Volts).

Temperature - temperature at which the device is to operate (in Degrees Celsius). (Default = 27).

Parameters (definable within model file)

The following is a list of parameters that can be stored in the associated model file:

IS - saturation current (in Amps). (Default = 1.0e-14).

RS - ohmic resistance (in Ohms). (Default = 0).

N - emission coefficient (Default = 1).

TT - transit-time (in seconds). (Default = 0).

CJO - zero-bias junction capacitance (in Farads). (Default = 0).

VJ - junction potential (in Volts). (Default = 1).

M - grading coefficient (Default = 0.5).

EG - activation energy (in eV). (Default = 1.11).

XTI - saturation-current temp exp. (Default = 3.0).

KF - flicker noise coefficient (Default = 0)

AF - flicker noise exponent (Default = 1).

FC - coefficient for forward-bias depletion capacitance formula (Default = 0.5).

BV - reverse breakdown voltage (in Volts). (Default = infinite).

IBV - current at breakdown voltage (in Amps). (Default = 1.0e-3).

TNOM - parameter measurement temperature (in °C) - If no value is specified, the default value assigned to TNOM on the SPICE Options page of the *Analyses Setup* dialog will be used (Default = 27).

Notes

The value for the `Initial Voltage` only applies if the **Use Initial Conditions** option is enabled on the **Transient/Fourier Analysis Setup** page of the *Analyses Setup* dialog.

The `Area Factor` affects the following three model parameters:

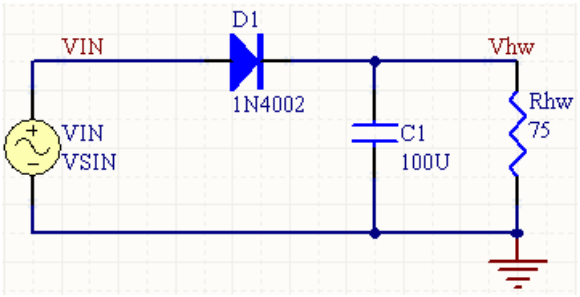
- saturation current (`IS`)
- ohmic resistance (`RS`)
- zero-bias junction capacitance (`CJO`)

If the `Area Factor` is omitted, a value of `1.0` is assumed.

The link to the required model file (`*.mdl`) is specified on the **Model Kind** tab of the *Sim Model* dialog. The Model Name is used in the netlist to reference this file.

Where a parameter has an indicated default (as part of the SPICE model definition), that default will be used if no value is specifically entered. The default should be applicable to most simulations. Generally you do not need to change this value.

Examples



Consider the diode in the above image, with the following characteristics:

- Pin1 (anode) is connected to net `VIN`
- Pin2 (cathode) is connected to net `Vhw`
- Designator is `D1`
- The linked simulation model file is `1N4002.mdl`.

If no values are entered for the parameters in the *Sim Model* dialog, the entries in the SPICE netlist would be:

```
*Schematic Netlist:
D1 VIN VHW 1N4002
.
.
*Models and Subcircuit:
.MODEL 1N4002 D(IS=2.55E-9 RS=0.042 N=1.75 TT=5.76E-6 CJO=1.85E-11 VJ=0.75
+ M=0.333 BV=100 IBV=1E-5 )
```

and the SPICE engine would use the indicated parameter information defined in the model file, along with default parameter values inherent to the model for those parameters not specified in the file.

If the following parameter values were specified on the **Parameters** tab of the *Sim Model* dialog:

- **Area Factor** = `3`
- **Initial Voltage** = `2`
- **Temperature** = `22`

then the entries in the SPICE netlist would be:

```
*Schematic Netlist:
D1 VIN VHW 1N4002 3 IC=2 TEMP=22
.
.
*Models and Subcircuit:
.MODEL 1N4002 D(IS=2.55E-9 RS=0.042 N=1.75 TT=5.76E-6 CJO=1.85E-11 VJ=0.75
```

```
+ M=0.333 BV=100 IBV=1E-5 )
```

In this case, the SPICE engine would use this information, in conjunction with the indicated parameters defined in the model file (and any defaults for parameters not specified).

PSpice Support

To make this device model compatible with PSpice, the following additional model parameters are supported and can be entered into a linked model file (`*.mdl`) for the device:

IBVL - low-level reverse breakdown knee current (in Amps). (Default = 0).

IKF - high-injection knee current (in Amps). (Default = infinite).

ISR - recombination current parameter (in Amps). (Default = 0).

NBV - reverse breakdown ideality factor. (Default = 1).

NBVL - low-level reverse breakdown ideality factor. (Default = 1).

NR - emission coefficient for isr. (Default = 2).

TBV1 - bv temperature coefficient – linear (in $^{\circ}C^{-1}$). (Default = 0).

TBV2 - bv temperature coefficient – quadratic (in $^{\circ}C^{-2}$). (Default = 0).

TIKF - ikf temperature coefficient – linear (in $^{\circ}C^{-1}$). (Default = 0).

TRS1 - rs temperature coefficient – linear (in $^{\circ}C^{-1}$). (Default = 0).

TRS2 - rs temperature coefficient – quadratic (in $^{\circ}C^{-2}$). (Default = 0).

Where a parameter has an indicated default, that default will be used if no value is specifically entered.

The format for the PSpice model file is:

`.MODEL` *ModelName* `D(`*Model Parameters*`)`,

where

- *ModelName* is the name of the model, the link to which is specified on the **Model Kind** tab of the *Sim Model* dialog. This name is used in the netlist (`@MODEL`) to reference the required model in the linked model file.
- *Model Parameters* are a list of supported parameters for the model, entered with values as required.

The following parameters – common to most devices in PSpice – are not supported:

T_ABS

T_MEASURED

T_REL_GLOBAL

T_REL_LOCAL.

For an example of using a PSpice-compatible diode model in a simulation, refer to the example project `Diode.PrjPCB`, which can be found in the `\Examples\Circuit Simulation\PSpice Examples\Diode` folder of the installation.

Inductor

Model Kind

General

Model Sub-Kind

Inductor

SPICE Prefix

L

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @VALUE ?"INITIAL CURRENT"|IC=@"INITIAL CURRENT"|
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

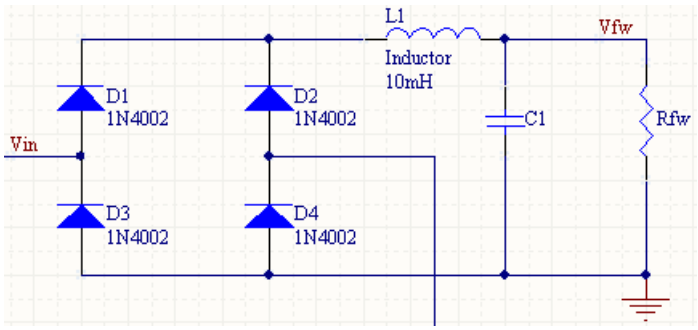
Value - value for the inductance (in Henrys).

Initial Current - time-zero current flowing through inductor (in Amps).

Notes

The value for the `Initial Current` only applies if the **Use Initial Conditions** option is enabled on the **Transient/Fourier Analysis Setup** page of the *Analyses Setup* dialog.

Examples



Consider the inductor in the above image, with the following characteristics:

- Pin1 (positive) is connected to net `Vin`
- Pin2 (negative) is connected to net `Vfw`
- Designator is `L1`
- Value = `10mH`.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
L1 Vin Vfw 10mH
```

PSpice Support

The existing Spice3f5 model for the Inductor device has been enhanced to support the general PSpice model form:

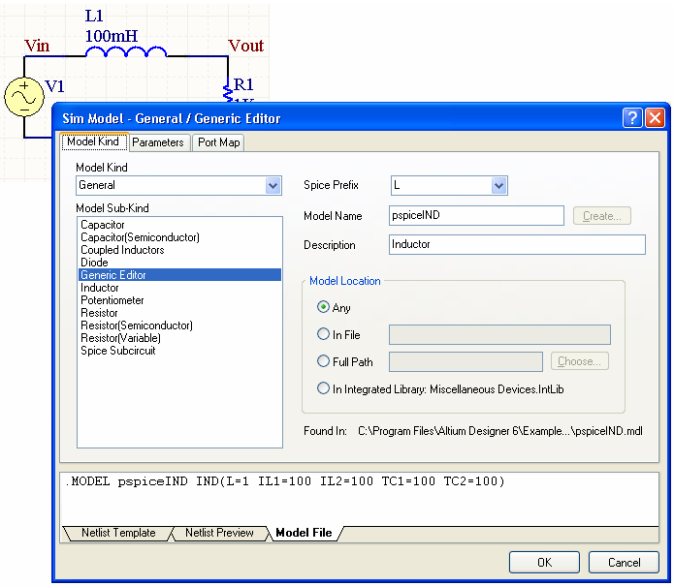
```
L<name> <(+) node> <(-) node> [model name]
<value> [IC = <initial value>]
```

A PSpice model of this type should be linked to a schematic component using a model file. Simply specify the model in a model file (`*.mdl`) then, in the *Sim Model* dialog, set the **Model Kind** to `General` and the **Model Sub-Kind** to `Generic Editor`. The Netlist Template Format should then be entered as follows:

```
@DESIGNATOR %1 %2 @VALUE @MODEL ?"INITIAL
CURRENT"|IC=@"INITIAL CURRENT"|
```

The value for the `INITIAL CURRENT` parameter is entered on the **Parameters** tab of the *Sim Model* dialog.

The netlist format for a PSpice Inductor model is specified using the Generic Editor due to the fact that the Spice3f5 Inductor model does not support use of a linked model file.



For the circuit to be parsed correctly, ensure that the **Spice Prefix** field is set to `L`.

In the **Model Name** field, enter the name specified for the model in the model file. Use the options in the Model Location region of the dialog to point to the required file. Click on the **Model File** tab to view the content of the model file.

The following additional model parameters are supported and can be entered into a linked model file (`*.mdl`) for the device:

L - inductance multiplier. (Default = 1).

IL1 - linear current coefficient (in Amp^{-1}). (Default = 0).

IL2 - quadratic current coefficient (in Amp^{-2}). (Default = 0).

TC1 - linear temperature coefficient (in $^{\circ}C^{-1}$). (Default = 0).

TC2 - quadratic temperature coefficient (in $^{\circ}C^{-2}$). (Default = 0).

Where a parameter has an indicated default, that default will be used if no value is specifically entered.

The following parameters – common to most devices in PSpice – are not supported:

T_ABS

T_MEASURED

T_REL_GLOBAL

T_REL_LOCAL.

The format for the PSpice model file is:

`.MODEL` *ModelName* `IND(`*Model Parameters*`)`,

where

- *ModelName* is the name of the model, the link to which is specified on the **Model Kind** tab of the *Sim Model* dialog. This name is used in the netlist (`@MODEL`) to reference the required model in the linked model file.
- *Model Parameters* are a list of supported parameters for the model, entered with values as required.

For an example of using a PSpice-compatible inductor model in a simulation, refer to the example project `Inductor.PrjPCB`, which can be found in the `\Examples\Circuit Simulation\PSpice Examples\Inductor` folder of the installation.

Potentiometer

Model Kind

General

Model Sub-Kind

Potentiometer

SPICE Prefix

R

SPICE Netlist Template Format

```
@"DESIGNATOR"A %1 %2 {@VALUE * @"SET POSITION"}
@"DESIGNATOR"B %2 %3 {@VALUE - (@VALUE * @"SET POSITION")}
```

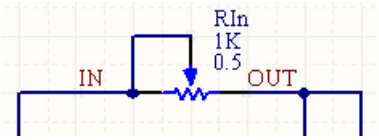
Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Value - value for the resistance (in Ohms).

Set Position - the position of the wiper along the resistors' track. The value can be in the range 0 (fully left/anti-clockwise) to 1 (fully right/clockwise), with 0.5 being the halfway point, with equal resistance on both sides.

Examples



Consider the potentiometer in the image above, with the following characteristics:

- Pin1 (Top) is connected to net `OUT`
- Pin2 (Bot) is connected to net `IN`
- Pin3 (Tap or wiper) is connected to net `IN`
- Designator is `RIn`
- Value = `1K`
- Set Position = `0.5`.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
RInA OUT IN {1K * 0.5}
RInB IN IN {1K - (1K * 0.5)}
```

Resistor

Model Kind

General

Model Sub-Kind

Resistor

SPICE Prefix

R

SPICE Netlist Template Format

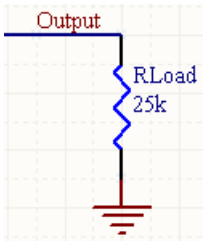
```
@DESIGNATOR %1 %2 @VALUE
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Value - value for the resistance (in Ohms).

Examples



Consider the resistor in the above image, with the following characteristics:

- Pin1 (Top) is connected to net `Output`
- Pin2 (Bot) is connected to net `GND`
- Designator is `RLoad`
- Value = `25k`.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
```

`RLoad OUTPUT 0 25k` - the ground net is always defined as `0` in the netlist.

Resistor (Semiconductor)

Model Kind

General

Model Sub-Kind

Resistor(Semiconductor)

SPICE Prefix

R

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 &VALUE &MODEL ?LENGTH|L=@LENGTH| ?WIDTH|W=@WIDTH|
?TEMPERATURE|TEMP=@TEMPERATURE|
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Value - value for the resistance (in Ohms).

Length - length of the resistor (in meters).

Width - width of the resistor (in meters). (Default = 1e-6).

Temperature - temperature at which the device is to operate (in Degrees Celsius). (Default = 27°C).

Parameters (definable within model file)

The following is a list of process-related parameters that can be stored in the associated model file:

TC1 - first order temperature coefficient (in Ohms/°C). (Default = 0).

TC2 - second order temperature coefficient (in Ohms/°C^2). (Default = 0).

RSH - sheet resistance (in Ohms).

DEFW - default width (in meters). - this value will be overridden by a value entered for `Width` in the *Sim Model* dialog.

NARROW - narrowing due to side etching (in meters). (Default = 0).

TNOM - parameter measurement temperature (in °C) - If no value is specified, the default value assigned to TNOM on the SPICE Options page of the *Analyses Setup* dialog will be used (Default = 27).

Notes

You can specify either a direct value for the resistance OR enter values for the resistors' length and width. In the case of the latter, a value for the resistance will be calculated, in conjunction with parameter information stored in the model.

The equation used to calculate the resistance from geometric data is:

```
R = RSH * ((L - NARROW) / (W - NARROW))
```

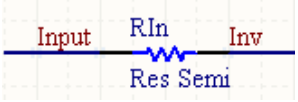
If a direct value for resistance is not specified, the model name and length must be supplied in order for the geometric-based resistance value to be calculated. If either the length or sheet resistance (`RSH`) is not specified, the default resistance value of `1K` will be used and a warning will be generated.

The link to the required model file (`*.mdl`) is specified on the **Model Kind** tab of the *Sim Model* dialog. The Model Name is used in the netlist to reference this file.

Entering a direct value for the resistance will override the geometric definition.

Where a parameter has an indicated default (as part of the SPICE model definition), that default will be used if no value is specifically entered. The default should be applicable to most simulations. Generally you do not need to change this value.

Examples



Consider the semiconductor resistor in the above image, with the following characteristics:

- Pin1 is connected to net `Input`
- Pin2 is connected to net `Inv`
- Designator is `RIn`
- The linked simulation model file is `RES.mdl`.

If a value for the resistance was entered directly, say `10K`, and no other parameters were specified on the **Parameters** tab of the *Sim Model* dialog, then the entry in the SPICE netlist would be:

```
*Schematic Netlist:
RIn Input Inv RES 10K
```

Consider now, instead of entering a direct value for the resistance, the following parameters were defined in the *Sim Model* dialog:

- **Length** = `10e-3`
- **Width** = `4e-3`
- **Temperature** = `24`.

The entry in the netlist would be:

```
*Schematic Netlist:
RIn INPUT INV RES L=10e-3 W=4e-3 TEMP=24
```

As long as a value for the sheet resistance (`RSH`) has been defined in the model file (`RES.mdl`), the value for the resistance will be calculated accurately from the geometric data given.

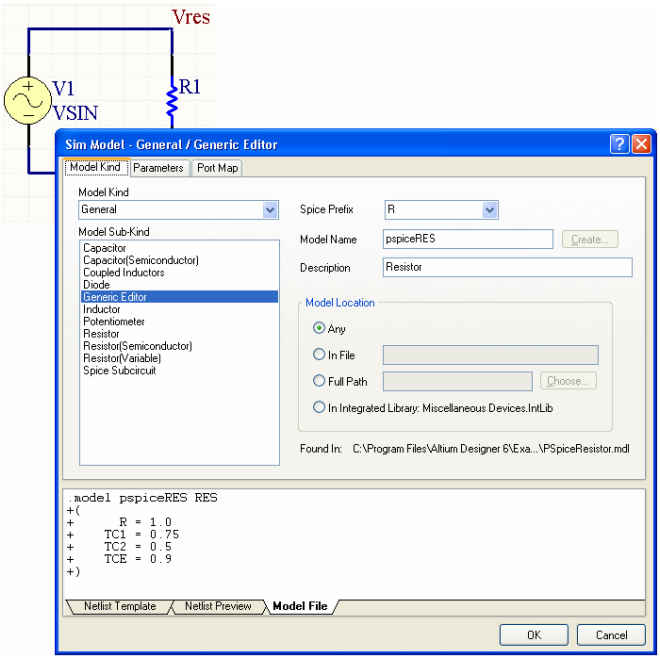
PSpice Support

The existing Spice3f5 model for the Resistor (Semiconductor) device has been enhanced to support the general PSpice model form:

```
R<name> <(+) node> <(-) node> [model name] <value> [TC = <TC1> [,<TC2>]]
```

A PSpice model of this type should be linked to a schematic component using a model file. Simply specify the model in a model file (`*.mdl`) then, in the *Sim Model* dialog, set the **Model Kind** to `General` and the **Model Sub-Kind** to `Generic Editor`. The Netlist Template Format should then be entered as follows:

```
@DESIGNATOR %1 %2 &MODEL &VALUE
?TC1/TC=@TC1?TC2|, @TC2| /
```



Although you could use the Spice3f5 Resistor (Semiconductor) model – as this model type allows use of a linked model file – specification of the netlist format for a PSpice Resistor model using the Generic Editor allows you to make use of the additional PSpice parameters (`[TC = <TC1> [,<TC2>]]`).

For the circuit to be parsed correctly, ensure that the **Spice Prefix** field is set to `R`.

In the **Model Name** field, enter the name specified for the model in the model file. Use the options in the Model Location region of the dialog to point to the required file. Click on the **Model File** tab to view the content of the model file.

The following additional model parameters are supported and can be entered into a linked model file (`*.mdl`) for the device:

R - resistance multiplier. (Default = 1).

TC1 - linear temperature coefficient (in $^{\circ}C^{-1}$). (Default = 0).

TC2 - quadratic temperature coefficient (in $^{\circ}C^{-2}$). (Default = 0).

TCE - exponential temperature coefficient (in %/$^{\circ}$C). (Default = 0).

Values for TC1 and TC2 can be entered on the **Parameters** tab of the *Sim Model* dialog. Where a parameter has an indicated default, that default will be used if no value is specifically entered – either on the **Parameters** tab or in the linked model file.

The format for the PSpice model file is:

`.MODEL` *ModelName* `RES(`*Model Parameters*`)`,

where

- *ModelName* is the name of the model, the link to which is specified on the **Model Kind** tab of the *Sim Model* dialog. This name is used in the netlist (`&MODEL`) to reference the required model in the linked model file.
- *Model Parameters* are a list of supported parameters for the model, entered with values as required.

The following parameters – common to most devices in PSpice – are not supported:

T_ABS

T_MEASURED

T_REL_GLOBAL

T_REL_LOCAL.

For an example of using a PSpice-compatible capacitor model in a simulation, refer to the example project `Resistor.PrjPCB`, which can be found in the `\Examples\Circuit Simulation\PSpice Examples\Resistor` folder of the installation.

Resistor (Variable)

Model Kind

General

Model Sub-Kind

Resistor(Variable)

SPICE Prefix

R

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 {@VALUE * @"SET POSITION"}
```

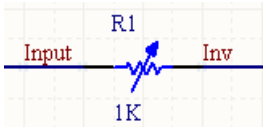
Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Value value for the resistance (in Ohms).

Set Position - the position of the wiper along the resistors' track. The value can be in the range 0 (fully left/anti-clockwise) to 1 (fully right/clockwise), with 0.5 being the halfway point (i.e. the resistors' value will be half that specified in the `Value` field).

Examples



Consider the variable resistor in the image above, with the following characteristics:

- Pin1 is connected to net `Input`
- Pin2 is connected to net `Inv`
- Designator is `R1`
- Value = `1K`
- Set Position = `0.5`.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
R1 INPUT INV 5E+2
```

Transistors

Bipolar Junction Transistor (BJT)

Model Kind

Transistor

Model Sub-Kind

BJT

SPICE Prefix

Q

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 @MODEL &"AREA FACTOR" &"STARTING CONDITION" ?"INITIAL B-E
VOLTAGE"|IC=@"INITIAL B-E VOLTAGE", @"INITIAL C-E VOLTAGE"| ?TEMPERATURE|TEMP=@TEMPERATURE|
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Area Factor - specifies the number of equivalent parallel devices of the specified model. This setting affects a number of parameters in the model.

Starting Condition - set to `OFF` to set terminal voltages to zero during operating point analysis. Can be useful as an aid in convergence.

Initial B-E Voltage - time-zero voltage across base-emitter terminals (in Volts).

Initial C-E Voltage - time-zero voltage across collector-emitter terminals (in Volts).

Temperature - temperature at which the device is to operate (in Degrees Celsius). If no value is specified, the default value assigned to TEMP on the SPICE Options page of the *Analyses Setup* dialog will be used (Default = 27).

Parameters (definable within model file)

The following is a list of parameters that can be stored in the associated model file:

IS - transport saturation current (in Amps). (Default = 1.0e-16).

BF - ideal maximum forward beta (Default = 100).

NF - forward current emission coefficient (Default = 1).

VAF - forward Early voltage (in Volts). (Default = infinite).

IKF - corner for forward beta high current roll-off (in Amps). (Default = infinite).

ISE - B-E leakage saturation current (in Amps). (Default = 0).

NE - B-E leakage emission coefficient (Default = 1.5).

BR - ideal maximum reverse beta (Default = 1).

NR - reverse current emission coefficient (Default = 1).

VAR - reverse Early voltage (in Volts). (Default = infinite).

IKR - corner for reverse beta high current roll-off (in Amps). (Default = infinite).

ISC - B-C leakage saturation current (in Amps). (Default = 0).

NC - B-C leakage emission coefficient (Default = 2).

RB - zero bias base resistance (in Ohms). (Default = 0).

IRB - current where base resistance falls halfway to it minimum value (in Amps). (Default = infinite).

RBM - minimum base resistance at high currents (in Ohms). (Default = RB).

RE - emitter resistance (in Ohms). (Default = 0).

RC - collector resistance (in Ohms). (Default = 0).

CJE - B-E zero-bias depletion capacitance (in Farads). (Default = 0).

VJE - B-E built-in potential (in Volts). (Default = 0.75)

MJE - B-E junction exponential factor (Default = 0.33).

TF - ideal forward transit time (in seconds). (Default = 0).

XTF - coefficient for bias dependence of TF (Default = 0).

VTF - voltage describing VBC dependence of TF (in Volts). (Default = infinite).

ITF - high-current parameter for effect on TF (in Amps). (Default = 0).

PTF - excess phase at freq=1.0/(TF*2PI) Hz (in Degrees). (Default = 0).

CJC - B-C zero-bias depletion capacitance (in Farads). (Default = 0).

VJC - B-C built-in potential (in Volts). (Default = 0.75).

MJC - B-C junction exponential factor (Default = 0.33).

XCJC - fraction of B-C depletion capacitance connected to internal base node (Default = 1).

TR - ideal reverse transit time (in seconds). (Default = 0).

CJS - zero-bias collector-substrate capacitance (in Farads). (Default = 0).

VJS - substrate junction built-in potential (in Volts). (Default = 0.75).

MJS - substrate junction exponential factor (Default = 0).

XTB - forward and reverse beta temperature exponent (Default = 0).

EG - energy gap for temperature effect on IS (in eV). (Default = 1.11).

XTI - temperature exponent for effect on IS (Default = 3).

KF - flicker noise coefficient (Default = 0).

AF - flicker noise exponent (Default = 1).

FC - coefficient for forward-bias depletion capacitance formula (Default = 0.5).

TNOM - parameter measurement temperature (in °C) - If no value is specified, the default value assigned to TNOM on the SPICE Options page of the *Analyses Setup* dialog will be used (Default = 27).

Notes

The model for the BJT is an adaptation of the integral charge control model of Gummel and Poon. This enhanced version of the original Gummel-Poon model includes several effects at high bias levels. When certain parameters are not specified, the model automatically defaults to that of the simpler Ebers-Moll model.

Ground is used as the substrate node.

The values for `Initial B-E Voltage` and `Initial C-E Voltage` only apply if the **Use Initial Conditions** option is enabled on the **Transient/Fourier Analysis Setup** page of the *Analyses Setup* dialog.

The `Area Factor` affects the following model parameters:

- transport saturation current (`IS`)
- corner for forward beta high current roll-off (`IKF`)
- B-E leakage saturation current (`ISE`)
- corner for reverse beta high current roll-off (`IKR`)
- B-C leakage saturation current (`ISC`)
- zero bias base resistance (`RB`)

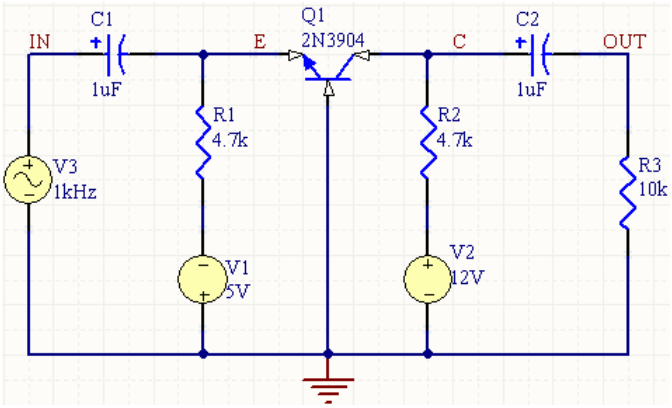
- current where base resistance falls halfway to its minimum value (`IRB`)
- minimum base resistance at high currents (`RBM`)
- emitter resistance (`RE`)
- collector resistance (`RC`)
- B-E zero-bias depletion capacitance (`CJE`)
- high-current parameter for effect on TF (`ITF`)
- B-C zero-bias depletion capacitance (`CJC`)
- zero-bias collector-substrate capacitance (`CJS`)

If the `Area Factor` is omitted, a value of `1.0` is assumed.

The link to the required model file (`*.mdl`) is specified on the **Model Kind** tab of the *Sim Model* dialog. The Model Name is used in the netlist to reference this file.

Where a parameter has an indicated default (as part of the SPICE model definition), that default will be used if no value is specifically entered. The default should be applicable to most simulations. Generally you do not need to change this value.

Examples



Consider the BJT in the above image, with the following characteristics:

- Pin1 (collector) is connected to net `C`
- Pin2 (base) is connected to net `GND`
- Pin3 (emitter) is connected to net `E`
- Designator is `Q1`
- The linked simulation model file is `2N3904.mdl`.

If no values are entered for the parameters in the *Sim Model* dialog, the entries in the SPICE netlist would be:

```
*Schematic Netlist:
Q1 C 0 E 2N3904
.
.
*Models and Subcircuit:
.MODEL 2N3904 NPN(IS=1.4E-14 BF=300 VAF=100 IKF=0.025 ISE=3E-13 BR=7.5 RC=2.4
+ CJE=4.5E-12 TF=4E-10 CJC=3.5E-12 TR=2.1E-8 XTB=1.5 KF=9E-16 )
```

and the SPICE engine would use the indicated parameter information defined in the model file, along with default parameter values inherent to the model for those parameters not specified in the file.

If the following parameter values were specified on the **Parameters** tab of the *Sim Model* dialog:

- **Area Factor** = `3`
- **Starting Condition** = `OFF`
- **Temperature** = `24`

then the entries in the SPICE netlist would be:

```
*Schematic Netlist:
```

```
Q1 C 0 E 2N3904 3 OFF  TEMP=24
.
.
*Models and Subcircuit:
.MODEL 2N3904 NPN(IS=1.4E-14 BF=300 VAF=100 IKF=0.025 ISE=3E-13 BR=7.5 RC=2.4
+ CJE=4.5E-12 TF=4E-10 CJC=3.5E-12 TR=2.1E-8 XTB=1.5 KF=9E-16 )
```

In this case, the SPICE engine would use this information, in conjunction with the indicated parameters defined in the model file (and any defaults for parameters not specified).

PSpice Support

Many of the parameters that can be included in a linked model file for this type of device are common to both Spice3f5 and PSpice. Those that are supported can be found in the previous section – *Parameters (definable within model file)*. The following PSpice-based parameters are *not supported* for this device type:

CN	-	quasi-saturation temperature coefficient for hole mobility
D	-	quasi-saturation temperature coefficient for scattering-limited hole carrier velocity
GAMMA	-	epitaxial region doping factor
ISS	-	substrate p-n saturation current
NK	-	high-current roll-off coefficient
NS	-	substrate p-n emission coefficient
QCO	-	epitaxial region charge factor
QUASIMOD	-	quasi-saturation model flag for temperature dependence
RCO	-	epitaxial region resistance
TRB1	-	RB temperature coefficient (linear)
TRB2	-	RB temperature coefficient (quadratic)
TRC1	-	RC temperature coefficient (linear)
TRC2	-	RC temperature coefficient (quadratic)
TRE1	-	RE temperature coefficient (linear)
TRE2	-	RE temperature coefficient (quadratic)
TRM1	-	RBM temperature coefficient (linear)
TRM2	-	RBM temperature coefficient (quadratic)
VG	-	quasi-saturation extrapolated bandgap voltage at 0° K
VO	-	carrier mobility knee voltage
XCJC2	-	fraction of CJC connected internally to Rb
XCJS	-	fraction of CJS connected internally to Rc

Junction Field-Effect Transistor (JFET)

Model Kind

Transistor

Model Sub-Kind

JFET

SPICE Prefix

J

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 @MODEL &"AREA FACTOR" &"STARTING CONDITION" ?"INITIAL D-S
VOLTAGE"|IC=@"INITIAL D-S VOLTAGE", @"INITIAL G-S VOLTAGE"| ?TEMPERATURE|TEMP=@TEMPERATURE|
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Area Factor - specifies the number of equivalent parallel devices of the specified model. This setting affects a number of parameters in the model.

Starting Condition - set to `OFF` to set terminal voltages to zero during operating point analysis. Can be useful as an aid in convergence.

Initial D-S Voltage - time-zero voltage across Drain-Source terminals (in Volts).

Initial G-S Voltage - time-zero voltage across Gate-Source terminals (in Volts).

Temperature - temperature at which the device is to operate (in Deg. C). If no value is specified, the default value assigned to TEMP on the SPICE Options page of the *Analyses Setup* dialog will be used (Default = 27).

Parameters (definable within model file)

The following is a list of parameters that can be stored in the associated model file:

VTO - threshold voltage V_{TO} (in Volts). (Default = -2.0).

BETA - transconductance parameter β (in A/V^2). (Default = 1.0e-4)

LAMBDA - channel-length modulation parameter λ (in 1/V). (Default = 0).

RD - drain ohmic resistance (in Ohms). (Default = 0).

RS - source ohmic resistance (in Ohms). (Default = 0).

CGS - zero-bias G-S junction capacitance C_{GS} (in Farads). (Default = 0).

CGD - zero-bias G-D junction capacitance C_{GD} (in Farads). (Default = 0).

PB - gate junction potential (in Volts). (Default = 1).

IS - gate junction saturation current I_S (in Amps). (Default = 1.0e-14).

B - doping tail parameter (Default = 1).

KF - flicker noise coefficient (Default = 0).

AF - flicker noise exponent (Default = 1).

FC - coefficient for forward-bias depletion capacitance formula (Default = 0.5).

TNOM - parameter measurement temperature (in °C) - If no value is specified, the default value assigned to TNOM on the SPICE Options page of the *Analyses Setup* dialog will be used (Default = 27).

Notes

The model for the JFET is based on the FET model of Shichman and Hodges.

The values for the `Initial D-S Voltage` and `Initial G-S Voltage` only apply if the **Use Initial Conditions** option is enabled on the **Transient/Fourier Analysis Setup** page of the *Analyses Setup* dialog.

The `Area Factor` affects the following model parameters:

- transconductance parameter (`BETA`)
- drain ohmic resistance (`RD`)
- source ohmic resistance (`RS`)

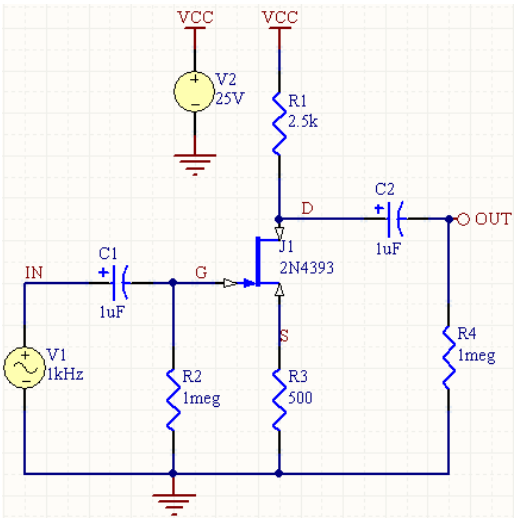
- zero-bias G-S junction capacitance (`CGS`)
- zero-bias G-D junction capacitance (`CGD`)
- gate junction saturation current (`IS`).

If the `Area Factor` is omitted, a value of `1.0` is assumed.

The link to the required model file (`*.mdl`) is specified on the **Model Kind** tab of the *Sim Model* dialog. The Model Name is used in the netlist to reference this file.

Where a parameter has an indicated default (as part of the SPICE model definition), that default will be used if no value is specifically entered. The default should be applicable to most simulations. Generally you do not need to change this value.

Examples



Consider the JFET in the above image, with the following characteristics:

- Pin1 (Drain) is connected to net `D`
- Pin2 (Gate) is connected to net `G`
- Pin3 (Source) is connected to net `S`
- Designator is `J1`
- The linked simulation model file is `2N4393.mdl`.

If no values are entered for the parameters in the *Sim Model* dialog, the entries in the SPICE netlist would be:

```
*Schematic Netlist:
J1 D G S 2N4393
.
.
*Models and Subcircuit:
.MODEL 2N4393 NJF(VTO=-1.422 BETA=0.009109 LAMBDA=0.006 RD=1 RS=1 CGS=4.06E-12
+ CGD=4.57E-12 IS=2.052E-13 KF=1.23E-16 )
```

and the SPICE engine would use the indicated parameter information defined in the model file, along with default parameter values inherent to the model for those parameters not specified in the file.

If the following parameter values were specified on the **Parameters** tab of the *Sim Model* dialog:

- **Area Factor** = `4`
- **Temperature** = `29`

then the entries in the SPICE netlist would be:

```
*Schematic Netlist:
J1 D G S 2N4393 4  TEMP=29
.
.
*Models and Subcircuit:
```

```
.MODEL 2N4393 NJF(VTO=-1.422 BETA=0.009109 LAMBDA=0.006 RD=1 RS=1 CGS=4.06E-12
+ CGD=4.57E-12 IS=2.052E-13 KF=1.23E-16 )
```

In this case, the SPICE engine would use this information, in conjunction with the indicated parameters defined in the model file (and any defaults for parameters not specified).

PSpice Support

To make this device model compatible with PSpice, the following additional model parameters are supported and can be entered into a linked model file (`*.mdl`) for the device:

ALPHA - ionization coefficient (in $Volt^{-1}$). (Default = 0).

BETATCE - BETA exponential temperature coefficient (in $Amp/Volt^2$). (Default = 1E-4).

ISR - gate p-n recombination current parameter (in Amps). (Default = 0).

M - gate p-n grading coefficient. (Default = 0.5).

N - gate p-n emission coefficient. (Default = 1).

NR - emission coefficient for isr. (Default = 2).

VK - ionization knee voltage (in Volts). (Default = 0).

VTOTC - VTO temperature coefficient (in Volt/˚C). (Default = 0).

XTI - IS temperature coefficient. (Default = 3).

Where a parameter has an indicated default, that default will be used if no value is specifically entered.

The following parameters – common to most devices in PSpice – are not supported:

T_ABS

T_MEASURED

T_REL_GLOBAL

T_REL_LOCAL.

The format for the PSpice model file is:

`.MODEL` *ModelName* `NJF(`*Model Parameters*`)` – N-channel JFET

`.MODEL` *ModelName* `PJF(`*Model Parameters*`)` – P-channel JFET

where

- *ModelName* is the name of the model, the link to which is specified on the **Model Kind** tab of the *Sim Model* dialog. This name is used in the netlist (`@MODEL`) to reference the required model in the linked model file.
- *Model Parameters* are a list of supported parameters for the model, entered with values as required.

For an example of using a PSpice-compatible diode model in a simulation, refer to the example project `JFET.PrjPCB`, which can be found in the `\Examples\Circuit Simulation\PSpice Examples\Jfet` folder of the installation.

Metal Semiconductor Field-Effect Transistor (MESFET)

Model Kind

Transistor

Model Sub-Kind

MESFET

SPICE Prefix

Z

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 @MODEL &"AREA FACTOR" &"STARTING CONDITION" ?"INITIAL D-S
VOLTAGE"|IC=@"INITIAL D-S VOLTAGE", @"INITIAL G-S VOLTAGE"|
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Area Factor - specifies the number of equivalent parallel devices of the specified model. This setting

affects a number of parameters in the model.

Starting Condition - set to `OFF` to set terminal voltages to zero during operating point analysis. Can be useful as an aid in convergence.

Initial D-S Voltage - time-zero voltage across Drain-Source terminals (in Volts).

Initial G-S Voltage - time-zero voltage across Gate-Source terminals (in Volts).

Parameters (definable within model file)

The following is a list of parameters that can be stored in the associated model file:

VTO - pinch-off voltage (in Volts). (Default = -2.0).

BETA - transconductance parameter β (in A/V^2). (Default = 1.0e-4)

B - doping tail extending parameter (in 1/V). (Default = 0.3).

ALPHA - saturation voltage parameter (in 1/V). (Default = 2).

LAMBDA - channel-length modulation parameter λ (in 1/V). (Default = 0).

RD - drain ohmic resistance (in Ohms). (Default = 0).

RS - source ohmic resistance (in Ohms). (Default = 0).

CGS - zero-bias G-S junction capacitance (in Farads). (Default = 0).

CGD - zero-bias G-D junction capacitance (in Farads). (Default = 0).

PB - gate junction potential (in Volts). (Default = 1).

KF - flicker noise coefficient (Default = 0).

AF - flicker noise exponent (Default = 1).

FC - coefficient for forward-bias depletion capacitance formula (Default = 0.5).

Notes

The model for the MESFET is based on the GaAs FET model of Statz et-al.

The values for the `Initial D-S Voltage` and `Initial G-S Voltage` only apply if the **Use Initial Conditions** option is enabled on the **Transient/Fourier Analysis Setup** page of the *Analyses Setup* dialog.

The `Area Factor` affects the following model parameters:

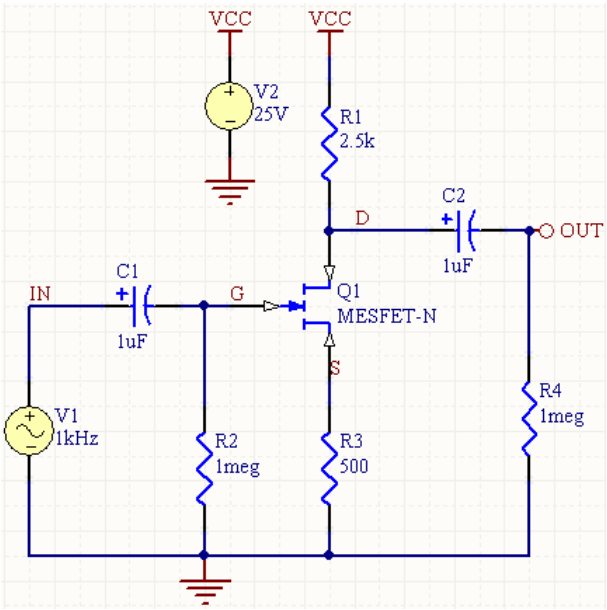
- transconductance parameter (`BETA`)
- doping tail extending parameter (`B`)
- saturation voltage parameter (`ALPHA`)
- drain ohmic resistance (`RD`)
- source ohmic resistance (`RS`)
- zero-bias G-S junction capacitance (`CGS`)
- zero-bias G-D junction capacitance (`CGD`).

If the `Area Factor` is omitted, a value of `1.0` is assumed.

The link to the required model file (`*.mdl`) is specified on the **Model Kind** tab of the *Sim Model* dialog. The Model Name is used in the netlist to reference this file.

Where a parameter has an indicated default (as part of the SPICE model definition), that default will be used if no value is specifically entered. The default should be applicable to most simulations. Generally you do not need to change this value.

Examples



Consider the MESFET in the above image, with the following characteristics:

- Pin1 (Drain) is connected to net `D`
- Pin2 (Gate) is connected to net `G`
- Pin3 (Source) is connected to net `S`
- Designator is `Q1`
- The linked simulation model file is `NMESFET.mdl`.

If no values are entered for the parameters in the *Sim Model* dialog, the entries in the SPICE netlist would be:

```
*Schematic Netlist:
ZQ1 D G S NMESFET
.
.
*Models and Subcircuit:
.MODEL NMESFET NMF(LEVEL=6, )
```

In this case, there are no parameter values specified in both the *Sim Model* dialog and the model file. The SPICE engine would therefore use the parameter default values inherent to the model.

Metal Oxide Semiconductor Field-Effect Transistor (MOSFET)

Model Kind

Transistor

Model Sub-Kind

MOSFET

SPICE Prefix

M

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %3 @MODEL ?LENGTH|L=@LENGTH| ?WIDTH|W=@WIDTH| ?"DRAIN AREA"|AD=@"DRAIN
AREA"| ?"SOURCE AREA"|AS=@"SOURCE AREA"| ?"DRAIN PERIMETER"|PD=@"DRAIN PERIMETER"| ?"SOURCE
PERIMETER"|PS=@"SOURCE PERIMETER"| ?NRD|NRD=@NRD| ?NRS|NRS=@NRS| &"STARTING CONDITION"
?"INITIAL D-S VOLTAGE"|IC=@"INITIAL D-S VOLTAGE", @"INITIAL G-S VOLTAGE", @"INITIAL B-S
VOLTAGE"| ?TEMPERATURE|TEMP=@TEMPERATURE|
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Length - channel length (in meters).

Width - channel width (in meters).

Drain Area - area of the Drain diffusion (in sq.meters).

Source Area - area of the Source diffusion (in sq.meters).

Drain Perimeter - perimeter of drain junction (in meters). (Default = 0).

Source Perimeter - perimeter of source junction (in meters). (Default = 0).

NRD - equivalent number of squares of the drain diffusion (Default = 1).

NRS - equivalent number of squares of the source diffusion (Default = 1).

Starting Condition - set to `OFF` to set terminal voltages to zero during operating point analysis. Can be useful as an aid in convergence.

Initial D-S Voltage - time-zero voltage across Drain-Source terminals (in Volts).

Initial G-S Voltage - time-zero voltage across Gate-Source terminals (in Volts).

Initial B-S Voltage - time-zero voltage across Bulk (substrate)-Source terminals (in Volts).

Temperature - temperature at which the device is to operate (in Degrees Celsius). If no value is specified, the default value assigned to TEMP on the SPICE Options page of the *Analyses Setup* dialog will be used (Default = 27).

M - multiplication factor. (Default = 1). This parameter is only available when using the EKV model.

Parameters (definable within model file)

The following is a list of parameters that can be stored in the associated model file, when using the Shichman-Hodges, MOS2, MOS3 or MOS6 models:

LEVEL - model index (Default = 1).

VTO - zero-bias threshold voltage V_{TO} (in Volts). (Default = 0).

KP - transconductance parameter (in A/V^2). (Default = 2.0e-5).

GAMMA - bulk threshold parameter γ (in $V^{1/2}$). (Default = 0).

PHI - surface potential φ (in Volts). (Default = 0.6).

LAMBDA - channel length modulation λ (in 1/V). This parameter is applicable to MOS1 and MOS2 model types only. (Default = 0).

RD - drain ohmic resistance (in Ohms). (Default = 0).

RS - source ohmic resistance (in Ohms). (Default = 0).

CBD - zero-bias B-D junction capacitance (in Farads). (Default = 0).

CBS - zero-bias B-S junction capacitance (in Farads). (Default = 0).

IS - bulk junction saturation current I_S (in Amps). (Default = 1.0e-14).

PB - bulk junction potential (in Volts). (Default = 0.8).

CGSO - Gate-Source overlap capacitance per meter channel width (in Farads per meter). (Default = 0).

CGDO - Gate-Drain overlap capacitance per meter channel width (in Farads per meter). (Default = 0).

CGBO - Gate-Bulk overlap capacitance per meter channel length (in Farads per meter). (Default = 0).

RSH - Drain and Source diffusion sheet resistance (in Ohms). (Default = 0).

CJ - zero-bias bulk junction bottom capacitance per square meter of junction area (in Farads/m^2). (Default = 0).

MJ - bulk junction bottom grading coefficient (Default = 0.5).

CJSW - zero-bias bulk junction sidewall capacitance per meter of junction perimeter (in Farads/meter). (Default = 0).

MJSW - bulk junction sidewall grading coefficient (Default = 0.5 - LEVEL1; 0.33 - LEVEL2,3).

JS - bulk junction saturation current per square meter of junction area (in Amps/m^2).

TOX - oxide thickness (in meters). (Default = 1.0e-7).

NSUB - substrate doping (in 1/cm^3). (Default = 0).

NSS - surface state density (in 1/cm^2). (Default = 0).

NFS - fast surface state density (in 1/cm^2). (Default = 0).

TPG - type of gate material:
+1 (default) = opposite to substrate
-1 = same as substrate
0 = Al gate

XJ - metallurgical junction depth (in meters). (Default = 0). This applies to Levels 2 (MOS2) and 3 (MOS3) only.

LD - lateral diffusion (in meters). (Default = 0).

UO - surface mobility (in cm^2/Vs). (Default = 600).

UCRIT - critical field for mobility degradation (in V/cm). This parameter is applicable to the MOS2 model only. (Default = 1.0e4).

UEXP - critical field exponent in mobility degradation. This parameter is applicable to the MOS2 model only. (Default = 0).

UTRA - transverse field coefficient (mobility). This parameter has been deleted with respect to the MOS2 model. (Default = 0).

VMAX - maximum drift velocity of carriers (in m/s). (Default = 0).

NEFF - total channel-charge (fixed and mobile) coefficient. This parameter is applicable to the MOS2 model only. (Default = 1).

KF - flicker noise coefficient (Default = 0).

AF - flicker noise exponent (Default = 1).

FC - coefficient for forward-bias depletion capacitance formula (Default = 0.5).

DELTA - width effect on threshold voltage. This parameter is applicable to MOS2 and MOS3 model types only. (Default = 0).

THETA - mobility modulation (in 1/V). This parameter is applicable to the MOS3 model only. (Default = 0).

ETA - static feedback. This parameter is applicable to the MOS3 model only. (Default = 0).

KAPPA - saturation field factor. This parameter is applicable to the MOS3 model only. (Default = 0.2).

TNOM - parameter measurement temperature (in °C) - If no value is specified, the default value assigned to TNOM on the SPICE Options page of the *Analyses Setup* dialog will be used (Default = 27).

The following is a list of parameters that can be stored in the associated model file, when using the BSIM or BSIM2 models:

LEVEL - model index (Default = 1).

VFB* - flat-band voltage (in Volts).

PHI* - surface inversion potential (in Volts).

K1* - body effect coefficient (in $V^{1/2}$).

K2* - Drain/Source depletion charge-sharing coefficient.

ETA* - zero-bias drain-induced barrier-lowering coefficient.

MUZ - zero-bias mobility (in cm^2/Vs).

DL - shortening of channel (in µm).

DW - narrowing of channel (in µm).

U0* - zero-bias transverse-field mobility degradation coefficient (in V^{-1}).

U1* - zer0-bias velocity saturation coefficient (in µm/V).

X2MZ* - sens. of mobility to substrate bias at V_{ds}=0 (in cm^2/V^2s).

X2E* - sens. of drain-induced barrier lowering effect to substrate bias (in V^{-1}).

X3E* - sens. of drain-induced barrier lowering effect to drain bias at $V_{ds}=V_{dd}$ (in V^{-1}).

X2U0* - sens. of transverse field mobility degradation effect to substrate bias (in V^{-2}).

X2U1* - sens. of velocity saturation effect to substrate bias (in μmV^{-2}).

MUS - mobility at zero substrate bias and at $V_{ds}=V_{dd}$ (in cm^2/V^2s).

X2MS* - sens. of mobility to substrate bias at $V_{ds}=V_{dd}$ (in cm^2/V^2s).

X3MS* - sens. of mobility to drain bias at $V_{ds}=V_{dd}$ (in cm^2/V^2s).

X3U1* - sens. of velocity saturation effect on drain bias at $V_{ds}=V_{dd}$ (in μmV^{-2}).

TOX - gate oxide thickness (in µm).

TEMP - temperature at which parameters were measured (in °C)

VDD - measurement bias range (in Volts).

CGDO - gate-drain overlap capacitance per meter channel width (in F/m).

CGSO - gate-source overlap capacitance per meter channel width (in F/m).

CGBO - gate-bulk overlap capacitance per meter channel length (in F/m).

XPART - gate-oxide capacitance-charge model flag.

N0* - zero-bias sub-threshold slope coefficient.

NB* - sens. of sub-threshold slope to substrate bias.

ND* - sens. of sub-threshold slope to drain bias.

RSH - drain and source diffusion sheet resistance (in Ohms).

JS - source drain junction current density (in A/m^2).

PB - built-in potential of source drain junction (in Volts).

MJ - grading coefficient of source drain junction.

PBSW - built-in potential of source drain junction sidewall (in Volts).

MJSW - grading coefficient of source drain junction sidewall.

CJ - source drain junction capacitance per unit area (in F/m^2).

CJSW - source drain junction sidewall capacitance per unit length (in F/m).

WDF - source drain junction default width (in meters).

DELL - source drain junction length reduction (in meters).

The following is a list of parameters that can be stored in the associated model file, when using the BSIM3 model:

LEVEL - model index (Default = 1).

MOBMOD - mobility model selector (Default = 1).

CAPMOD - flag for the short-channel capacitance model (Default = 2).

NQSMOD - flag for NQS model (Default = 0).

NOIMOD - flag for noise model (Default = 1).

VTH0 - threshold voltage (at V_{BS}=0) for Large L (in Volts). (Default = 0.7 – NMOS; -0.7 – PMOS). (See BSIM3-Related notes).

K1 - first-order body effect coefficient (in $V^{1/2}$). (Default = 0.5). (See BSIM3-Related notes).

K2 - second-order body effect coefficient (Default = 0). (See BSIM3-Related notes).

K3 - narrow width coefficient (Default = 80).

K3B - body effect coefficient of K3 (in 1/V). (Default = 0).

W0 - narrow width parameter (in meters) (Default = 2.5e-6).

NLX - lateral non-uniform doping parameter (in meters). (Default = 1.74e-7).

VBM - maximum applied body bias in Vth calculation (in Volts). (Default = -3.0).

DVT0 - first coefficient of short-channel effect on Vth (Default = 2.2).

DVT1 - second coefficient of short-channel effect on Vth (Default = 0.53).

DVT2 - body-bias coefficient of short-channel effect on Vth (in 1/V). (Default = -0.032).

DVT0W - first coefficient of narrow width effect on Vth for small channel length (in 1/m). (Default = 0).

DVT1W - second coefficient of narrow width effect on Vth for small channel length (in 1/m). (Default = 5.3e6).

DVT2W - body-bias coefficient of narrow width effect for small channel length (in 1/V). (Default = -0.032).

U0 - mobility at TEMP = TNOM (in $cm^2/V/s$). (Default = 670.0 – NMOSFET; 250.0 – PMOSFET).

UA - first-order mobility degradation coefficient (in m/V). (Default = 2.25e-9).

UB - second-order mobility degradation coefficient (in $(m/V)^2$). (Default = 5.87e-19).

UC - body-effect of mobility degradation coefficient.
For MOBMOD = 1 or 2, measured in m/V^2, Default = -4.65e-11
For MOBMOD = 3, measured in 1/V, Default = -0.046.

VSAT - saturation velocity at TEMP = TNOM (in m/sec). (Default = 8.0e4).

A0 - bulk charge effect coefficient for channel length (Default = 1.0).

AGS - gate bias coefficient of Abulk (in 1/V). (Default = 0).

B0 - bulk charge effect coefficient for channel width (in meters). (Default = 0).

B1 - bulk charge effect width offset (in meters). (Default = 0).

KETA - body-bias coefficient of bulk charge effect (in 1/V). (Default = -0.047).

A1 - first non-saturation effect parameter (in 1/V). (Default = 0).

A2 - second non-saturation factor (Default = 1).

RDSW - parisitic resistance per unit width (in Ω-μm^{WR}). (Default = 0).

PRWB - body effect coefficient of RDSW (in $V^{-1/2}$). (Default = 0).

PRWG - gate bias effect coefficient of RDSW (in 1/V). (Default = 0).

WR - width offset from Weff for RDS calculation (Default = 1).

WINT - width offset fitting parameter from I-V without bias (in meters). (Default = 0).

LINT - length offset fitting parameter from I-V without bias (in meters). (Default = 0).

DWG - coefficient of Weff's gate dependence (in m/V). (Default = 0).

DWB - coefficient of Weff's substrate body-bias dependence (in $m/V^{1/2}$). (Default = 0).

VOFF - offset voltage in the subthreshold region at large W and L (in Volts). (Default = -0.08).

NFACTOR - subthreshold swing factor (Default = 1).

ETA0 - DIBL coefficient in subthreshold region (Default = 0.08).

ETAB - body-bias coefficient for the subthreshold DIBL effect (in 1/V). (Default = -0.07).

DSUB - DIBL coefficient exponent in subthreshold region (Default = DROUT).

CIT - interface trap capacitance (in F/m^2). (Default = 0).

CDSC - Drain/Source to channel coupling capacitance (in F/m^2). (Default = 2.4e-4).

CDSCB - body-bias sensitivity of CDSC (in F/Vm^2). (Default = 0).

CDSCD - Drain-bias sensitivity of CDSC (in F/Vm^2). (Default = 0).

PCLM - channel length modulation parameter (Default = 1.3).

PDIBLC1 - first output resistance DIBL effect correction parameter (Default = 0.39).

PDIBLC2 - second output resistance DIBL effect correction parameter (Default = 0.0086).

PDIBLCB - body-effect coefficient of DIBL correction parameters (in 1/V). (Default = 0).

DROUT - L dependence coefficient of the DIBL correction parameter in Rout (Default = 0.56).

PSCBE1 - first substrate current body-effect parameter (in V/m). (Default = 4.24e8).

PSCBE2 - second substrate current body-effect parameter (in m/V). (Default = 1.0e-5).

PVAG - gate dependence of Early voltage (Default = 0).

DELTA - Effective Vds parameter (in Volts). (Default = 0.01).

NGATE - poly gate doping concentration (in cm^{-3}). (Default = 0).

ALPHA0 - the first parameter of impact ionization current (in m/V). (Default = 0).

BETA0 - the second parameter of impact ionization current (in Volts). (Default = 30).

RSH - source drain sheet resistance (in Ω/square). (Default = 0).

JS - source drain junction saturation current per unit area (in A/m^2). (Default = 1.0e-4).

XPART - charge partitioning rate flag (Default = 0).

CGSO - non LDD region source-gate overlap capacitance per channel length (in F/m). (See BSIM3-Related notes).

CGDO - non LDD region drain-gate overlap capacitance per channel length (in F/m). (See BSIM3-Related notes).

CGBO - gate bulk overlap capacitance per unit channel length (in F/m). (Default = 0).

CJ - bottom junction capacitance per unit area (in F/m^2). (Default = 5e-4).

MJ - bottom junction capacitance grating coefficient (Default = 0.5).

MJSW - Source/Drain side junction capacitance grading coefficient (Default = 0.33).

CJSW - Source/Drain side junction capacitance per unit area (in F/m). (Default = 5e-10).

CJSWG - Source/Drain gate sidewall junction capacitance grading coefficient (in F/m). (Default = CJSW).

MJSWG - Source/Drain gate sidewall junction capacitance coefficient. (Default = MJSW).

PBSW - Source/Drain side junction built-in potential (in Volts). (Default = 1.0).

PB - bottom built-in potential (in Volts). (Default = 1.0).

PBSWG - Source/Drain gate sidewall junction built-in potential (in Volts). (Default = PBSW).

CKAPPA - coefficient for lightly doped region overlap capacitance (in F/m). (Default = 0.6).

CF - Fringing field capacitance (in F/m). (See BSIM3-Related notes).

CLC - constant term for the short channel model (in meters). (Default = 0.1e-6).

CLE - exponential term for the short channel model (Default = 0.6).

DLC - length offset fitting parameter from C-V (in meters). (Default = LINT).

DWC - width offset fitting parameter from C-V (in meters). (Default = WINT).

ELM - Elmore constant of the channel (Default = 5).

WL - coefficient of length dependence for width offset (in m^{WLN}). (Default = 0).

WLN - power of length dependence for width offset (Default = 1.0).

WW - coefficient of width dependence for width offset (in m^{WWN}). (Default = 0).

WWN - power of width dependence for width offset (Default = 1.0).

WWL - coefficient of length and width cross term for width offset (in $m^{WWN+WLN}$). (Default = 0).

LL - coefficient of length dependence for length offset (in m^{LLN}). (Default = 0).

LLN - power of length dependence for length offset (Default = 1.0).

LW - coefficient of width dependence for length offset (in m^{LWN}). (Default = 0).

LWN - power of width dependence for length offset (Default = 1.0).

LWL - coefficient of length and width cross term for length offset (in $m^{LWN+LLN}$). (Default = 0).

TNOM - parameter measurement temperature (in °C) - If no value is specified, the default value assigned to TNOM on the SPICE Options page of the *Analyses Setup* dialog will be used (Default = 27).

UTE - mobility temperature exponent (Default = -1.5).

KT1 - temperature coefficient for threshold voltage (in Volts). (Default = -0.11).

KT1L - channel length dependence of the temperature coefficient for threshold voltage (in V*m). (Default = 0).

KT2 - body-bias coefficient of Vth temperature effect (Default = 0.022).

UA1 - temperature coefficient for UA (in m/V). (Default = 4.31e-9).

UB1 - temperature coefficient for UB (in $(m/V)^2$). (Default = -7.61e-18).

UC1 - temperature coefficient for UC.
For MOBMOD = 1 or 2, measured in m/V^2, Default = -5.6e-11
For MOBMOD = 3, measured in 1/V, Default = -0.056.

AT - temperature coefficient for saturation velocity (in m/s). (Default = 3.3e4).

PRT - temperature coefficient for RDSW (in Ω-µm). (Default = 0).

NJ - emission coefficient of junction (Default = 1).

XTI - junction current temperature exponent coefficient (Default = 3.0).

NOIA - noise parameter A (Default = 1e20 – NMOS; 9.9e18 – PMOS).

NOIB - noise parameter B (Default = 5e4 – NMOS; 2.4e3 – PMOS).

NOIC - noise parameter C (Default = -1.4e-12 – NMOS; 1.4e-12 – PMOS).

EM - saturation field (in V/m). (Default = 4.1e7).

AF - frequency exponent (Default = 1).

EF - flicker exponent (Default = 1).

KF - flicker noise parameter (Default = 0).

TOX - gate oxide thickness (in meters). (Default = 1.5e-8).

XJ - junction depth (in meters). (Default = 1.5e-7).

GAMMA1 - body-effect coefficient near the surface (in $V^{1/2}$). (See BSIM3-Related notes).

GAMMA2 - body-effect coefficient in the bulk (in $V^{1/2}$). (See BSIM3-Related notes).

NCH - channel doping concentration (in $1/cm^3$). (Default = 1.7e17). (See BSIM3-Related notes).

NSUB - substrate doping concentration (in $1/cm^3$). (Default = 6e16).

VBX - Vbs at which the depletion region width equals XT (in Volts). (See BSIM3-Related notes).

XT - doping depth (in meters). (Default = 1.55e-7).

LMIN - minimum channel length (in meters). (Default = 0).

LMAX - maximum channel length (in meters). (Default = 1.0).

WMIN - minimum channel width (in meters). (Default = 0).

WMAX - maximum channel width (in meters). (Default = 1.0).

BINUNIT - Bin unit scale selector (Default = 1).

The following is a list of parameters that can be stored in the associated model file, when using the EKV model:

Process Related Parameters

COX - gate oxide capacitance per unit area (in F/m^2). (Default = 0.7e-3).

XJ - junction depth (in meters). (Default = 0.1e-6).

DW - channel width correction (in meters). (Default = 0).

DL - channel length correction (in meters). (Default = 0).

Basic Intrinsic Model Parameters

VTO - long-channel threshold voltage (in Volts). (Default = 0.5).

GAMMA - body effect parameter (in $V^{1/2}$). (Default = 1.0).

PHI - bulk Fermi potential (*2) (in Volts). (Default = 0.7).

KP - transconductance parameter (in A/V^2). (Default = 50.0e-6).

E0 (EO) - mobility reduction coefficient (in V/m). (Default = 1.0e12).

UCRIT - longitudinal critical field (in V/m). (Default = 2.0e6).

Optional Parameters

TOX - oxide thickness (in meters).

NSUB - channel doping (in cm^{-3}).

VFB - flat-band voltage (in Volts).

UO - low-field mobility (in cm^2/Vs).

VMAX - saturation velocity (in m/s).

THETA - mobility reduction coefficient (in 1/V). (Default = 0).

Channel Length Modulation and Charge Sharing Parameters

LAMBDA - depletion length coefficient (channel length modulation). (Default = 0.5).

WETA - narrow-channel effect coefficient. (Default = 0.25).

LETA - short-channel effect coefficient. (Default = 0.1).

Reverse Short-Channel Effect Parameters

Q0 (QO) - reverse short-channel effect peak charge density (in As/m^2). (Default = 0).

LK - reverse short-channel effect characteristic length (in meters). (Default = 0.29e-6).

Impact Ionization Related Parameters

IBA - first impact ionization coefficient (in 1/m). (Default = 0).

IBB - second impact ionization coefficient (in V/m). (Default = 3.0e8).

IBN - saturation voltage factor for impact ionization. (Default = 1.0).

Intrinsic Model Temperature Parameters

TCV - threshold voltage temperature coefficient (in V/K). (Default = 1.0e-3).

BEX - mobility temperature exponent. (Default = -1.5).

UCEX - longitudinal critical field temperature exponent. (Default = 0.8).

IBBT - temperature coefficient for IBB (in 1/K). (Default = 9.0e-4).

Flicker Noise Parameters

KF - flicker noise coefficient. (Default = 0).

AF - flicker noise exponent. (Default = 1).

Setup Parameters

NQS - non-quasi-static (NQS) operation switch. (Default = 0).

SATLIM - ratio defining the saturation limit. (Default = exp(4)).

Additional Parameters

LEVEL - model index (Default = 1).

TNOM - parameter measurement temperature (in °C) - If no value is specified, the default value assigned to TNOM on the SPICE Options page of the *Analyses Setup* dialog will be used (Default = 27).

IS - bulk junction saturation current (in Amps). (Default = 1.0e-14).

JS - bulk junction saturation current per square meter of junction area (in $Amps/m^2$).

JSW - sidewall saturation current per unit length (in A/m). (Default = 0).

N - bulk p-n emission coefficient. (Default = 1).

CBD - zero-bias B-D junction capacitance (in Farads). (Default = 0).

CBS - zero-bias B-S junction capacitance (in Farads). (Default = 0).

CJ - zero-bias bulk junction bottom capacitance per square meter of junction area (in Farads/m^2). (Default = 0).

CJSW - zero-bias bulk junction sidewall capacitance per meter of junction perimeter (in Farads/meter). (Default = 0).

MJ - bulk junction bottom grading coefficient. (Default = 0.5).

MJSW - bulk junction sidewall grading coefficient. (Default = 0.33).

FC - coefficient for forward-bias depletion capacitance formula. (Default = 0.5).

PB - bulk junction potential (in Volts). (Default = 0.8).

PBSW - built-in potential of source drain junction sidewall (in Volts). (Default = 1).

TT - bulk p-n transit time (in seconds). (Default = 0).

CGSO - Gate-Source overlap capacitance per meter channel width (in F/m). (Default = 0).

CGDO - Gate-Drain overlap capacitance per meter channel width (in F/m). (Default = 0).

CGBO - Gate-Bulk overlap capacitance per meter channel length (in F/m). (Default = 0).

RD - drain ohmic resistance (in Ohms). (Default = 0).

RS - source ohmic resistance (in Ohms). (Default = 0).

RSH - Drain and Source diffusion sheet resistance (in Ohms). (Default = 0).

RSC - source contact resistance (in Ohms). (Default = 0).

RDC - drain contact resistance (in Ohms). (Default = 0).

XTI - drain, source junction current temperature exponent. (Default = 0).

TR1 - first-order temperature coefficient for drain, source series resistance (in $°C^{-1}$). (Default = 0).

TR2 - second-order temperature coefficient for drain, source series resistance (in $°C^{-1}$). (Default = 0).

ACM - area calculation model. (Default = 0).

CJGATE - zero-bias gate-edge sidewall junction capacitance (in F/m). If no value is specified, the value assigned to CJSW will be used. This parameter is only used when ACM = 3. It is ignored otherwise.

GEO - shared geometry parameter. (Default = 0). This parameter is only used when ACM = 3. It is ignored otherwise.

HDIF - length of heavily doped diffusion (in m). (Default = 0). This parameter is only used when ACM = 2 or 3. It is ignored otherwise.

LD - lateral diffusion into channel (in m). (Default = 0).

LDIF - length of lightly doped diffusion near gate (in m). (Default = 0).

SCALM - model scaling factor. (Default = 1).

UPDATE - selects effective drain and source resistance model. (Default = 0). This parameter is only used when ACM = 1. It is ignored otherwise.

WMLT - width diffusion layer shrink reduction factor. (Default = 1).

Notes

General

The Simulator supports the following MOSFET device models, which differ only in their formulation of the I-V characteristic:

- Shichman-Hodges (`LEVEL`=1)
- MOS2 (`LEVEL`=2)
- MOS3 (`LEVEL`=3)
- BSIM (`LEVEL`=4)

- BSIM2 (LEVEL=5)
- MOS6 (LEVEL=6)
- BSIM3 (LEVEL=7)
- EKV (LEVEL=8).

The LEVEL parameter is used to specify which model to use. It is declared at the start of the parameter values list, entered in the associated model file. If no LEVEL parameter is declared, the default Schichman-Hodges model will be used.

The Bulk (substrate) node is connected, by default, to the Source node.

If any of the component-level Length, Width, Drain Area or Source Area parameters are not specified, default values will be used.

The values for the component-level NRD and NRS parameters are used to multiply the sheet resistance (RSH), in order to obtain an accurate representation of the parasitic series drain and source resistance of each transistor.

The values for the component-level parameters Initial D-S Voltage, Initial G-S Voltage and Initial B-S Voltage only apply if the **Use Initial Conditions** option is enabled on the **Transient/Fourier Analysis Setup** page of the *Analyses Setup* dialog.

The component-level Temperature parameter applies to LEVEL 1,2,3,6 & 8 MOSFET models (not BSIM type models).

The link to the required model file (*.mdl) is specified on the **Model Kind** tab of the *Sim Model* dialog. The Model Name is used in the netlist to reference this file.

Where a parameter has an indicated default (as part of the SPICE model definition), that default will be used if no value is specifically entered. The default should be applicable to most simulations. Generally you do not need to change this value.

BSIM/BSIM2-Related

The BSIM and BSIM2 models are designed to be used with a process characterization system. This system is responsible for providing all parameter information (values) automatically through the use of a process file and therefore no default parameter values are specified. As a consequence, all parameters are required to be specified and the absence of any will result in an error.

If the XPART parameter is set to 0, a 40/60 drain/source charge partition in saturation is selected. If this parameter is set to 1, a 0/100 drain/source charge partition is selected.

Certain model parameters (those marked with an asterisk in the BSIM/BSIM2 list of parameters) also have corresponding parameters dependent on length and width. For more information on these and other aspects of the MOSFET models, consult the SPICE reference manual.

BSIM3-Related

The following charge partition schemes are supported, selectable based on the value entered for the XPART parameter:

- XPART = 0 – a 0/100 drain/source charge partition os selected
- XPART = 0.5 – a 50/50 drain/source charge partition os selected
- XPART = 1 – a 40/60 drain/source charge partition os selected.

If no value is specified for one of the following parameters, it will be calculated:

- VTH0
- K1
- K2
- CGSO
- CGDO
- CF
- GAMMA1
- GAMMA2
- NCH
- VBX

For details of the calculations involved, refer to the BSIM3v3 User Manual.

The following BSIM3 model parameters are not supported in Altium Designer:

JSSW - side wall saturation current density.

CGS1 - light doped source-gate region overlap capacitance.

CGD1 - light doped drain-gate region overlap capacitance.

VFB - flat-band volatge parameter.

EKV-Related

The EKV MOSFET model was developed by the Electronics Laboratory (LEG) of the Swiss Federal Institute of Technology (EPFL). The model used in Altium Designer is version 2.6.

The following EKV model parameters are not supported in Altium Designer:

M or **NP** - parallel multiple device number.

N or **NS** - series multiple device number.

AVTO - area related threshold voltage mismatch parameter.

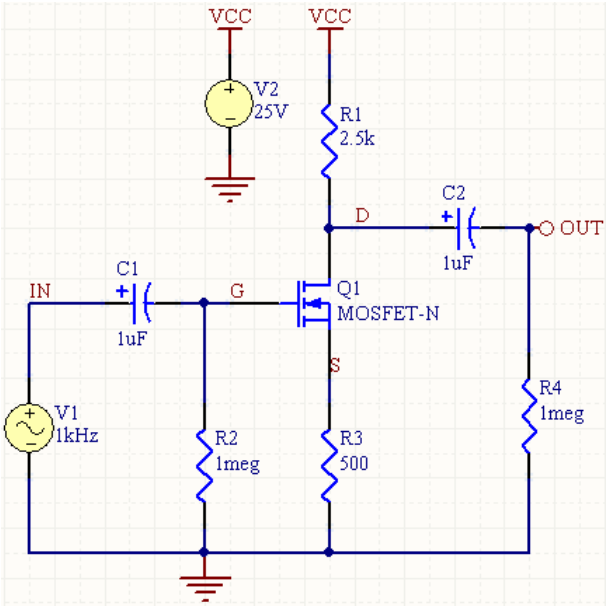
AKP - area related gain mismatch parameter.

AGAMMA - area related body effect mismatch parameter.

XQC - charge/capacitance model selector.

For more detailed information on model equations associated with the EKV MOSFET, refer to the document "*The EPFL-EKV MOSFET Model Equations for Simulation*", available from the following website - *http://legwww.epfl.ch/ekv/model.html*.

Examples



Consider the MOSFET in the above image, with the following characteristics:

- Pin1 (Drain) is connected to net `D`
- Pin2 (Gate) is connected to net `G`
- Pin3 (Source) is connected to net `S`
- The substrate node (Bulk) is connected to Pin3 (the Source node).
- Designator is `Q1`
- The linked simulation model file is `NMOS3.mdl`.

If no values are entered for the parameters in the *Sim Model* dialog, the entries in the SPICE netlist would be:

```
*Schematic Netlist:
MQ1 D G S S NMOS3
.
.
*Models and Subcircuit:
```

```
.MODEL NMOS3 NMOS(LEVEL=3)
```

In this case, there are no parameter values specified in the *Sim Model* dialog. In the model file, there is only the `LEVEL` parameter specified - corresponding to the use of the MOS3 model. The default values for all other parameters - inherent to the model - will be used.

PSpice Support

Of the existing MOSFET device models, the following are not supported with respect to PSpice compatibility:

- BSIM3 model version 2.0

For the other supported MOSFET device models, many of the parameters that can be included in a linked model file are common to both Spice3f5 and PSpice. Those that are supported can be found in the previous section – *Parameters (definable within model file)*. The following PSpice-based parameters are *not supported* for this device type.

GDSNOI - channel shot noise coefficient (use with NLEV=3)

JSSW - bulk p-n saturation sidewall current/length

L - channel length

N - bulk p-n emission coefficient

NLEV - noise equation selector

PBSW - bulk p-n sidewall potential

RB - bulk ohmic resistance

RDS - drain-source shunt resistance

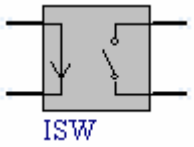
RG - gate ohmic resistance

TT - bulk p-n transit time

W - channel width

Switches

Current Controlled Switch



Model Kind

Switch

Model Sub-Kind

Current-Controlled

SPICE Prefix

W

SPICE Netlist Template Format

```
V@DESIGNATOR %1 %2 0V
@DESIGNATOR %3 %4 V@DESIGNATOR @MODEL &"INITIAL CONDITION"
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Initial Condition - the starting point for the switch, either open (OFF) or closed (ON).

Parameters (definable within model file)

The following is a list of parameters that can be stored in the associated model file:

IT - threshold current (in Amps). (Default = 0).

IH - hysteresis current (in Amps). (Default = 0).

RON - ON resistance (in Ohms). (Default = 1).

ROFF - OFF resistance (in Ohms). (Default = 1/`GMIN`). `GMIN` is an advanced SPICE option that sets the minimum conductance (maximum resistance) of any device in the circuit. It is specified on the **Spice Options** page of the *Analyses Setup* dialog and its default value is 1.0e-12 (mhos).

Notes

The model allows an almost ideal switch to be described. With careful selection of the ON and OFF resistances, they can effectively be seen as zero and infinity respectively, in comparison with other elements in the circuit.

The use of an ideal, highly non-linear element such as a switch can cause large discontinuities to occur in the circuit node voltages. The rapid state change caused by opening and closing a switch can cause numerical round off or tolerance problems, leading to time step difficulties, or erroneous results. When using switches, take the following precautions:

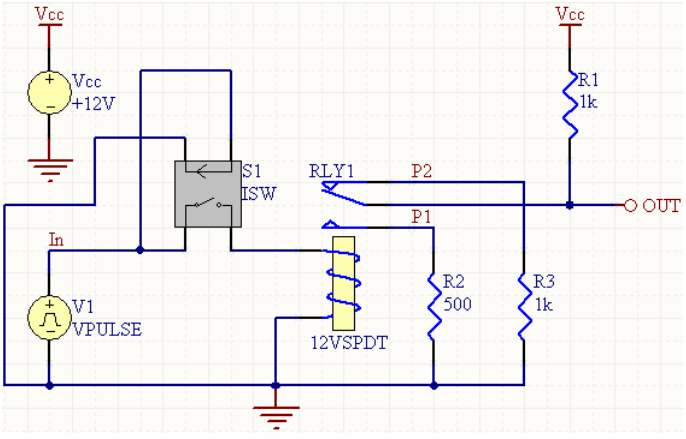
- Set switch impedances (`RON` and `ROFF`) just high or low enough to be negligible with respect to other elements.
- When modeling real devices such as MOSFETS, set the on resistance to a realistic level for the size of the device being modeled.
- If a wide range of ON to OFF resistance must be used (`ROFF/RON >1e+12`), then the error tolerance during transient analysis should be decreased. Set the `TRTOL` parameter on the **Spice Options** page of the *Analyses Setup* dialog to `1`.
- When a switch is placed around a capacitor, then the `CHGTOL` parameter should also be reduced (try `1e-16`).

The link to the required model file (`*.mdl`) is specified on the **Model Kind** tab of the *Sim Model* dialog. The Model Name is used in the netlist to reference this file.

Where a parameter has an indicated default (as part of the SPICE model definition), that default will be used if no value is specifically entered. The default should be applicable to most simulations. Generally you do not need to change this value.

The simulation-ready current controlled switch component (`ISW`) can be found in the Simulation Special Function integrated library (`\Library\Simulation\Simulation Special Function.IntLib`).

Examples



Consider the current controlled switch in the above image, with the following characteristics:

- Pin1 (positive controlling node) is connected to net `IN`
- Pin2 (negative controlling node) is connected to net `GND`
- Pin3 (positive output node) is connected to net `NetRLY1_4` (pin 4 of RLY1).
- Pin4 (negative output node) is connected to net `IN`
- Designator is `S1`
- Initial Condition of switch is `OFF` (open contact).
- The linked simulation model file is `ISW.mdl`.

The entries in the SPICE netlist would be:

```
*Schematic Netlist:
VWS1 NetRLY1_4 IN 0V
WS1 IN 0 VWS1 ISW OFF
.
.
*Models and Subcircuit:
.MODEL ISW CSW()
```

The SPICE engine would use the value for the `Initial Condition` specified on the **Parameters** tab of the *Sim Model* dialog. As there are no parameter values specified in the model file, the engine will use the default values for all other parameters.

PSpice Support

To make this device model compatible with PSpice, the following additional model parameters are supported and can be entered into a linked model file (`*.mdl`) for the device:

IOFF - control current for OFF state (in Amps). (Default = 0).

ION - control current for ON state (in Amps). (Default = 1E-3).

Where a parameter has an indicated default, that default will be used if no value is specifically entered.

The format for the PSpice model file is:

`.MODEL` *ModelName* `ISWITCH(`*Model Parameters*`)`,

where

- *ModelName* is the name of the model, the link to which is specified on the **Model Kind** tab of the *Sim Model* dialog. This name is used in the netlist (`@MODEL`) to reference the required model in the linked model file.
- *Model Parameters* are a list of supported parameters for the model, entered with values as required.

The following parameters – common to most devices in PSpice – are not supported:

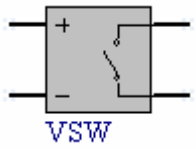
T_ABS

T_MEASURED

T_REL_GLOBAL

T_REL_LOCAL.

Voltage Controlled Switch



Model Kind

Switch

Model Sub-Kind

Voltage-Controlled

SPICE Prefix

S

SPICE Netlist Template Format

```
@DESIGNATOR %3 %4 %1 %2 @MODEL &"INITIAL CONDITION"
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Initial Condition - the starting point for the switch, either open (OFF) or closed (ON).

Parameters (definable within model file)

The following is a list of parameters that can be stored in the associated model file:

VT - threshold voltage (in Volts). (Default = 0).

VH - hysteresis voltage (in Volts). (Default = 0).

RON - ON resistance (in Ohms). (Default = 1).

ROFF - OFF resistance (in Ohms). (Default = 1/`GMIN`). `GMIN` is an advanced SPICE option that sets the minimum conductance (maximum resistance) of any device in the circuit. It is specified on the **Spice Options** page of the *Analyses Setup* dialog and its default value is 1.0e-12 (mhos).

Notes

The model allows an almost ideal switch to be described. With careful selection of the ON and OFF resistances, they can effectively be seen as zero and infinity respectively, in comparison with other elements in the circuit.

The use of an ideal, highly non-linear element such as a switch can cause large discontinuities to occur in the circuit node voltages. The rapid state change caused by opening and closing a switch can cause numerical round off or tolerance problems, leading to time step difficulties, or erroneous results. When using switches, take the following precautions:

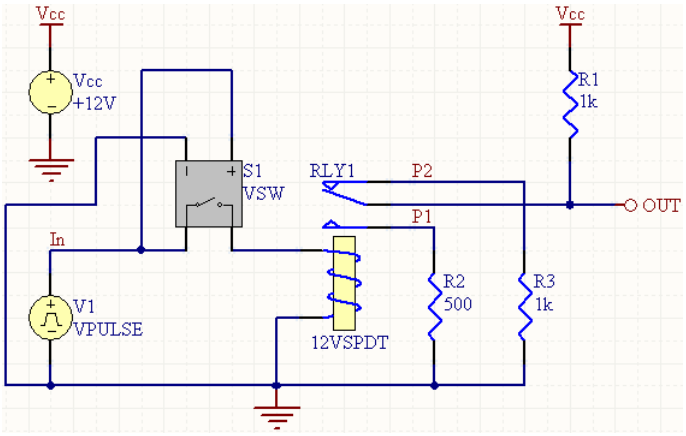
- Set switch impedances (`RON` and `ROFF`) just high or low enough to be negligible with respect to other elements.
- When modeling real devices such as MOSFETS, set the on resistance to a realistic level for the size of the device being modeled.
- If a wide range of ON to OFF resistance must be used (`ROFF/RON >1e+12`), then the error tolerance during transient analysis should be decreased. Set the `TRTOL` parameter on the **Spice Options** page of the *Analyses Setup* dialog to `1`.
- When a switch is placed around a capacitor, then the `CHGTOL` parameter should also be reduced (try `1e-16`).

The link to the required model file (`*.mdl`) is specified on the **Model Kind** tab of the *Sim Model* dialog. The Model Name is used in the netlist to reference this file.

Where a parameter has an indicated default (as part of the SPICE model definition), that default will be used if no value is specifically entered. The default should be applicable to most simulations. Generally you do not need to change this value.

The simulation-ready voltage controlled switch component (`VSW`) can be found in the Simulation Special Function integrated library (`\Library\Simulation\Simulation Special Function.IntLib`).

Examples



Consider the voltage controlled switch in the above image, with the following characteristics:

- Pin1 (positive controlling node) is connected to net `IN`
- Pin2 (negative controlling node) is connected to net `GND`
- Pin3 (positive output node) is connected to net `NetRLY1_4` (pin 4 of RLY1).
- Pin4 (negative output node) is connected to net `IN`
- Designator is `S1`
- Initial Condition of switch is `OFF` (open contact).
- The linked simulation model file is `VSW.mdl`.

The entries in the SPICE netlist would be:

```
*Schematic Netlist:
S1 NetRLY1_4 IN IN 0 VSW OFF
.
.
*Models and Subcircuit:
.MODEL VSW SW()
```

The SPICE engine would use the value for the `Initial Condition` specified on the **Parameters** tab of the *Sim Model* dialog. As there are no parameter values specified in the model file, the engine will use the default values for all other parameters.

PSpice Support

To make this device model compatible with PSpice, the following additional model parameters are supported and can be entered into a linked model file (`*.mdl`) for the device:

VOFF - control voltage for OFF state (in Volts). (Default = 0).

VON - control voltage for ON state (in Volts). (Default = 1).

Where a parameter has an indicated default, that default will be used if no value is specifically entered.

The format for the PSpice model file is:

`.MODEL` *ModelName* `VSWITCH(`*Model Parameters*`)`,

where

- *ModelName* is the name of the model, the link to which is specified on the **Model Kind** tab of the *Sim Model* dialog. This name is used in the netlist (`@MODEL`) to reference the required model in the linked model file.
- *Model Parameters* are a list of supported parameters for the model, entered with values as required.

The following parameters – common to most devices in PSpice – are not supported:

T_ABS

T_MEASURED

T_REL_GLOBAL

T_REL_LOCAL.

For an example of using a PSpice-compatible voltage-controlled switch model in a simulation, refer to the example project `PSpice Switch.PrjPCB`, which can be found in the `\Examples\Circuit Simulation\PSpice Examples\PSpice switch` folder of the installation.

Transmission Lines

Lossless Transmission Line

Model Kind

Transmission Line

Model Sub-Kind

Lossless

SPICE Prefix

T

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 Z0=@"CHAR. IMPEDANCE" ?"TRANSMISSION DELAY"/TD=@"TRANSMISSION
DELAY"//F=@FREQUENCY ?"NORMALISED LENGTH"|NL=@"NORMALISED LENGTH"|/ ?"INITIAL VOLTAGE
1"|IC=@"INITIAL VOLTAGE 1", @"INITIAL CURRENT 1", @"INITIAL VOLTAGE 2", @"INITIAL CURRENT 2"|
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Char. Impedance - characteristic impedance (in Ohms). (Default = 50).

Transmission Delay - Transmission delay (in seconds). (Default = 10n).

Frequency	-	frequency (in Hertz).
Normalised Length	-	normalized electrical length of the transmission line with respect to the wavelength in the line, at the specified frequency.
Port 1 Voltage	-	time-zero voltage at port 1 of the transmission line (in Volts).
Port 1 Current	-	time-zero current at port 1 of the transmission line (in Amps).
Port 2 Voltage	-	time-zero voltage at port 2 of the transmission line (in Volts).
Port 2 Current	-	time-zero current at port 2 of the transmission line (in Amps).

Notes

The length of the line must be expressed in one of the following two ways:

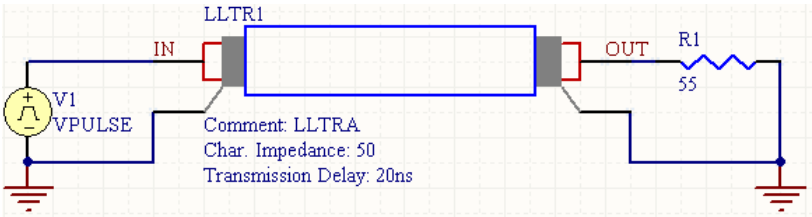
- the `Transmission Delay` is specified directly (eg, `TD=10ns`)
- a value for the `Frequency` is specified, together with a value for the `Normalised Length`.

If a value for `Frequency` is specified but a value for the `Normalised Length` is omitted, then `0.25` is assumed (that is, the frequency is assumed to be the quarter-wave frequency).

The values for Port 1 and Port 2 Initial Voltages and Currents only apply if the **Use Initial Conditions** option is enabled on the **Transient/Fourier Analysis Setup** page of the *Analyses Setup* dialog.

The simulation-ready lossless transmission line component (`LLTRA`) can be found in the Simulation Transmission Line integrated library (`\Library\Simulation\Simulation Transmission Line.IntLib`).

Examples



Consider the lossless transmission line in the above image, with the following characteristics:

- Pin1 (positive node of Port 1) is connected to net `IN`
- Pin2 (negative node of Port 1) is connected to net `GND`
- Pin3 (positive node of Port 2) is connected to net `OUT`
- Pin4 (negative node of Port 2) is connected to net `GND`
- Designator is `LLTR1`
- Char. Impedance = `50Ohms`
- Transmission Delay = `20ns`.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
TLLTR1 IN 0 OUT 0 Z0=50 TD=20NS
```

Lossy Transmission Line

Model Kind

Transmission Line

Model Sub-Kind

Lossy

SPICE Prefix

O

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component-level)

It is not possible to pass parameter information directly from the schematic component. The parameters must be defined in the associated model file.

Parameters (definable within model file)

The following is a list of parameters that can be stored in the associated model file:

R	-	resistance per unit length (in Ohms/unit). (Default = 0).
L	-	inductance per unit length (in Henrys/unit). (Default = 0).
G	-	conductance per unit length (in mhos/unit). (Default = 0).
C	-	capacitance per unit length (in Farads/unit). (Default = 0).
LEN	-	length of transmission line.
REL	-	breakpoint control (in arbitrary units). (Default = 1).
ABS	-	breakpoint control (in arbitrary units). (Default = 1).
NOSTEPLIMIT	-	a flag that, when set, will remove the restriction of limiting time-steps to less than the line delay. (Default = not set).
NOCONTROL	-	a flag that, when set, prevents limiting of the time-step, based on convolution error criteria. (Default = not set).
LININTERP	-	a flag that, when set, will use linear interpolation instead of the default quadratic interpolation, for calculation of delayed signals. (Default = not set).
MIXEDINTERP	-	a flag that, when set, uses a metric for determining whether quadratic interpolation is applicable and, if it isn't, uses linear interpolation. (Default = not set).
COMPACTREL	-	a specific quantity used to control the compaction of past history values used for convolution. By default, this quantity uses the value specified for the relative simulation error tolerance (`RELTOL`), which is defined on the **Spice Options** page of the *Analyses Setup* dialog.
COMPACTABS	-	a specific quantity used to control the compaction of past history values used for convolution. By default, this quantity uses the value specified for the absolute current error tolerance (`ABSTOL`), which is defined on the **Spice Options** page of the *Analyses Setup* dialog.
TRUNCNR	-	a flag that, when set, turns on the use of the Newton-Raphson iteration method to determine an appropriate time-step in the time-step control routines. (Default = not set, whereby a trial and error method is used - cutting the previous time-step in half each time).
TRUNCDONTCUT	-	a flag that, when set, removes the default cutting of the time-step to limit errors in the actual calculation of impulse-response related quantities. (Default = not set).

Notes

The operation of the lossy transmission line model (LTRA) is based on the convolution of the transmission line's impulse responses with its inputs.

The length of the transmission line (`LEN`) must be specified.

Setting any of the `NOCONTROL`, `NOSTEPLIMIT` and `TRUNCDONTCUT` flags increases simulation speed but may affect the accuracy of the results.

Using larger values for `COMPACTREL` and `COMPACTABS` will result in reduced accuracy but greater simulation speed. If defined, these parameters will only be applied if the `TRYTOCOMPACT` option is enabled on the **Spice Options** page of the *Analyses Setup* dialog.

The link to the required model file (`*.mdl`) is specified on the **Model Kind** tab of the *Sim Model* dialog. The Model Name is used in the netlist to reference this file.

Where a parameter has an indicated default (as part of the SPICE model definition), that default will be used if no value is specifically entered in the model file. The default should be applicable to most simulations. Generally you do not need to change this value.

A lossy transmission line with zero loss may be more accurate than the lossless transmission line due to implementation details.

The simulation-ready lossy transmission line component (`LTRA`) can be found in the Simulation Transmission Line integrated library (`\Library\Simulation\Simulation Transmission Line.IntLib`).

You can easily create and reference your own model file. To do this copy the file `ltra.MDL`. Edit this new model file and change the string immediately after the `.MODEL` statement to be the same as the new file name, then edit the parameters as required.

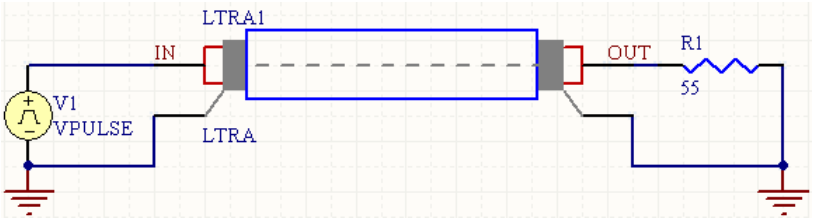
For example, from the existing model file `ltra.MDL`:

```
.MODEL LTRA LTRA(R=0.000 L=9.130n C=3.650p LEN=1.000)
```

You could create a new file, `ltra10.MDL`:

```
.MODEL LTRA10 LTRA(R=0.2 L=32n C=13p LEN=10.000)
```

Examples



Consider the lossy transmission line in the above image, with the following characteristics:

- Pin1 (positive node of Port 1) is connected to net `IN`
- Pin2 (negative node of Port 1) is connected to net `GND`
- Pin3 (positive node of Port 2) is connected to net `OUT`
- Pin4 (negative node of Port 2) is connected to net `GND`
- Designator is `LTRA1`
- The linked simulation model file is `LTRA.mdl`.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
OLTRA1 IN 0 OUT 0 LTRA
.
.
*Models and Subcircuit:
.MODEL LTRA LTRA(R=0.000 L=9.130N C=3.650P LEN=1.000)
```

The SPICE engine would use the indicated parameter information defined in the model file, along with default parameter values inherent to the model for those parameters not specified in the file.

Uniform Distributed RC (lossy) Transmission Line

Model Kind

Transmission Line

Model Sub-Kind

Uniform Distributed RC

SPICE Prefix

U

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 @MODEL L=@LENGTH ?"NO. SEGMENTS"|N=@"NO. SEGMENTS"|
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Length - length of the RC line (in meters). (Default = 1).

No. Segments - the number of lumped segments to use in modeling the RC line. (Default = 6).

Parameters (definable within model file)

The following is a list of parameters that can be stored in the associated model file:

K - propagation constant. (Default = 2).

FMAX - maximum frequency of interest (in Hertz). (Default = 1.0G).

RPERL - resistance per unit length (in Ohms/m). (Default = 1000).

CPERL - capacitance per unit length (in Farads/m). (Default = 1.0e-15).

ISPERL - saturation current per unit length (in Amps/m). (Default = 0).

RSPERL - diode resistance per unit length (in Ohms/m). (Default = 0).

Notes

The model is derived from a model proposed by L. Gertzberrg. The model is accomplished by a subcircuit type expansion of the URC line into a network of lumped RC segments, with internally generated nodes. The RC segments are in geometric progression, increasing toward the middle of the line, with `K` as a proportionality constant. If a value for the number of lumped segments to be used in modeling the line is not specified, it will be calculated using the following formula:

```
N = (log[FMAX * (R/L) * (C/L) * 2 * Pi * L2 * ((K-1)/K)2]) / logK
```

The line consists of resistor and capacitor segments only, unless a non-zero value is assigned to the `ISPERL` parameter. In this case, the capacitors are replaced with reverse-biased diodes, possessing the following characteristics:

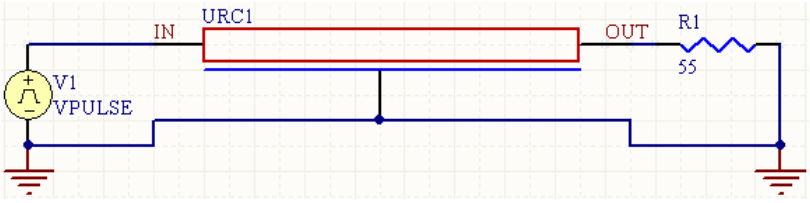
- a zero-bias junction capacitance equivalent to the capacitance replaced
- a saturation current of `ISPERL` Amps/m of transmission line
- an optional series resistance equivalent to `RSPERL` Ohms/m of transmission line.

The link to the required model file (`*.mdl`) is specified on the **Model Kind** tab of the *Sim Model* dialog. The Model Name is used in the netlist to reference this file.

For model file parameters, where a parameter has an indicated default (as part of the SPICE model definition), that default will be used if no value is specifically entered. The default should be applicable to most simulations. Generally you do not need to change this value.

The simulation-ready URC transmission line component (URC) can be found in the Simulation Transmission Line integrated library (`\Library\Simulation\Simulation Transmission Line.IntLib`).

Examples



Consider the URC transmission line in the above image, with the following characteristics:

- Pin1 (node 1) is connected to net `IN`
- Pin2 (node to which capacitances of the RC line are connected) is connected to net `GND`
- Pin3 (node 2) is connected to net `OUT`
- Designator is `URC1`
- Length = `1`

- No. Segments = 6
- The linked simulation model file is URC.mdl.

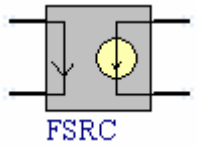
The entry in the SPICE netlist would be:

```
*Schematic Netlist:
UURC1 IN 0 OUT URC L=1 N=6
.
.
*Models and Subcircuit:
.MODEL URC URC()
```

In this case, there are no parameter values specified in the model file. The SPICE engine would therefore use the parameter values defined in the *Sim Model* dialog, along with the default values inherent to the model for all other parameters that are definable in the model file.

Current Sources

Current-Controlled Current Source



Model Kind

Current Source

Model Sub-Kind

Current-Controlled

SPICE Prefix

F

SPICE Netlist Template Format

```
V@DESIGNATOR %1 %2 0V
@DESIGNATOR %3 %4 V@DESIGNATOR @GAIN
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Gain - current gain of the source (in Amps).

Notes

This source produces a current at the output terminals that is a linear function of the current at the input terminals, dependant on the gain of the source.

The current-controlled current source actually implements two individual devices, as can be seen from the Netlist template. The first is a 0V voltage source, which acts as an ammeter, to measure the current input and then the actual current-controlled current source that references it. The direction of positive controlling current flow is from the positive node, through the source to the negative node of the 0V voltage source.

The characteristic equation for this source is:

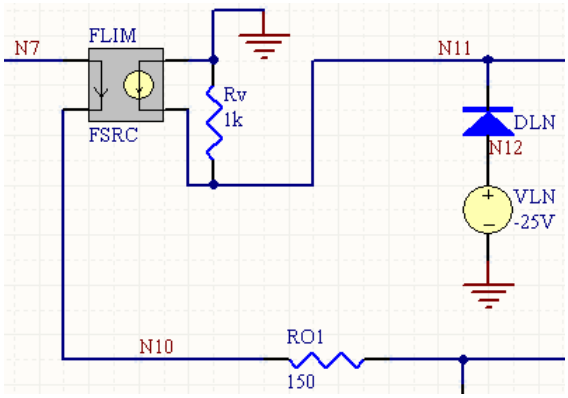
i = fi

where,

f is the current gain

The simulation-ready current controlled current source component (FSRC) can be found in the Simulation Sources integrated library (\Library\Simulation\Simulation Sources.IntLib).

Examples



Consider the current-controlled current source in the above image, with the following characteristics:

- Pin1 (positive controlling node) is connected to net `N7`
- Pin2 (negative controlling node) is connected to net `N10`
- Pin3 (positive output node) is connected to net `GND`
- Pin4 (negative output node) is connected to net `N11`
- Designator is `FLIM`
- Gain = `1`.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
VFLIM N7 N10 0V
FLIM 0 N11 VFLIM 1
```

PSpice Support

The following general PSpice model form is supported:

```
F<name> <(+) node> <(-) node> POLY(<value>) <controlling V device name> < <polynomial coefficient value> >
```

This device does not support linked model files. The netlist format for a PSpice model in the above form should be specified using the Generic Editor. In the *Sim Model* dialog, set the **Model Kind** to `General` and the **Model Sub-Kind** to `Generic Editor`.

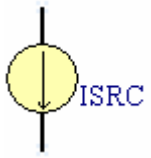
For the circuit to be parsed correctly, ensure that the **Spice Prefix** field is set to F.

The following example generic netlist template format could be used for this model type:

```
@DESIGNATOR %1 %2 POLY (@dimension) @ControlSource @coeffs
```

The values for the `dimension`, `ControlSource` and `coeffs` parameters are entered on the **Parameters** tab of the *Sim Model* dialog.

DC Current Source



Model Kind

Current Source

Model Sub-Kind

DC Source

SPICE Prefix

I

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @VALUE #"AC MAGNITUDE"|AC @"AC MAGNITUDE"| @"AC PHASE"
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Value	-	amplitude of the source current (in Amps).
AC Magnitude	-	the magnitude of the source when used in an AC Small Signal Analysis (typically 1A).
AC Phase	-	the phase of the source when used in an AC Small Signal Analysis.

Notes

This source produces a constant current output and is generally used to power the circuit.

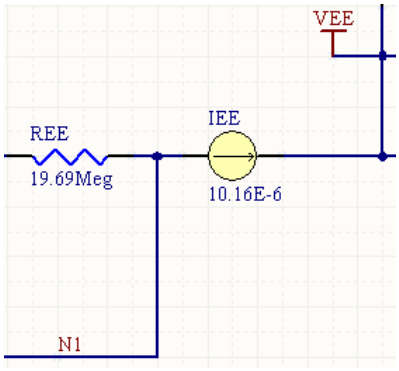
If a value for the DC source current is not specified, an error will occur when parsing the circuit to the Simulator.

If specifying AC criteria, the following should be observed;

- If a value for the `AC Magnitude` is entered, a value for the `AC Phase` *MUST* also be given, otherwise an error will occur when parsing the circuit.
- If a value for the `AC Magnitude` is omitted, but a value for `AC Phase` is defined, the circuit will parse to the Simulator OK, but the SPICE netlist will not contain any AC information for the source.

The simulation-ready DC current source component (`ISRC`) can be found in the Simulation Sources integrated library (`\Library\Simulation\Simulation Sources.IntLib`).

Examples



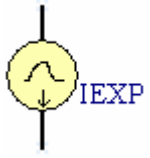
Consider the DC voltage source in the above image, with the following characteristics:

- Pin1 (positive) is connected to net `N1`
- Pin2 (negative) is connected to net `VEE`
- Designator is `IEE`
- Value = `10.16E-6`
- No AC parameters are specified.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
IEE N1 VEE 10.16E-6
```

Exponential Current Source



Model Kind

Current Source

Model Sub-Kind

Exponential

SPICE Prefix

I

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 ?"DC MAGNITUDE"|DC @"DC MAGNITUDE"| EXP(?"INITIAL VALUE"/&"INITIAL
VALUE"//0/ ?"PULSED VALUE"/&"PULSED VALUE"//5/ ?"RISE DELAY TIME"/&"RISE DELAY TIME"//1U/
?"RISE TIME CONSTANT"/&"RISE TIME CONSTANT"//700N/ ?"FALL DELAY TIME"/&"FALL DELAY TIME"//2U/
&"FALL TIME CONSTANT") #"AC MAGNITUDE"|AC @"AC MAGNITUDE"| @"AC PHASE"
```

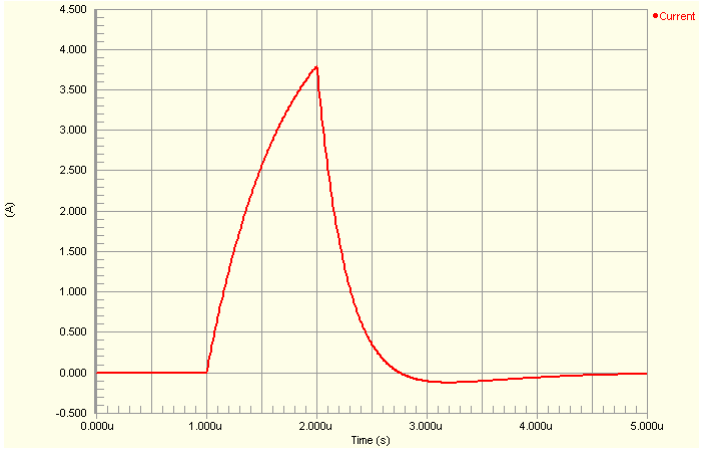
Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

DC Magnitude	-	DC offset used in an Operating Point Analysis. (Default = 0).
AC Magnitude	-	the magnitude of the source when used in an AC Small Signal Analysis. (Default = 1).
AC Phase	-	the phase of the source when used in an AC Small Signal Analysis. (Default = 0).
Initial Value	-	current amplitude at time zero (in Amps). (Default = 0).
Pulsed Value	-	maximum amplitude of the output swing (in Amps). (Default = 5).
Rise Delay Time	-	the point in time, from t_0, where the output begins to rise from the Initial Value to the Pulsed Value (in seconds). (Default = 1u).
Rise Time Constant	-	RC charging time constant (in seconds). (Default = 700n).
Fall Delay Time	-	the point in time, from t_0, where the output begins to fall from the Pulsed Value back to the Initial Value (in seconds). Must be >0. (Default = 2u).
Fall Time Constant	-	RC discharging time constant (in seconds). (Default = 300n).

Notes

Use this source to create a pulse current waveform with an exponential rising and/or falling edge.



The adjacent image shows an example waveform produced by an exponential current source (connected to a 1Ohm load) with the parameters set to their default values:

The shape of the waveform is described by the following formulae:

$I(t_0 \text{ to } t_{RD}) = I_{IV}$

$I(t_{RD} \text{ to } t_{FD}) = I_{IV} + (I_{PV} - I_{IV})\ (1 - e^{-(t-t_{RD})/t_{RT}})$

$I(t_{FD} \text{ to } t_{STOP}) = I_{IV} + (I_{PV} - I_{IV})\ (-e^{-(t-t_{RD})/t_{RT}}) + (I_{IV} - I_{PV})\ (1 - e^{-(t-t_{FD})/t_{FT}})$

where,

t is an instance of time

I_{IV} is the initial value of the current

I_{PV} is the pulsed value of the current

t_{RD} is the Rise Delay

t_{RT} is the Rise Time

t_{FD} is the Fall Delay and

t_{FT} is the Fall Time.

The simulation-ready exponential current source component (`IEXP`) can be found in the Simulation Sources integrated library (`\Library\Simulation\Simulation Sources.IntLib`).

Frequency Modulated Sinusoidal Current Source

Model Kind

Current Source

Model Sub-Kind

Single-Frequency FM

SPICE Prefix

I

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 ?"DC MAGNITUDE"|DC @"DC MAGNITUDE"| SFFM(@OFFSET @AMPLITUDE @"CARRIER FREQUENCY" @"MODULATION INDEX" @"SIGNAL FREQUENCY") #"AC MAGNITUDE"|AC @"AC MAGNITUDE"| @"AC PHASE"
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

DC Magnitude - DC offset used in an Operating Point Analysis. (Default = 0).

AC Magnitude - the magnitude of the source when used in an AC Small Signal Analysis. (Default = 1).

AC Phase - the phase of the source when used in an AC Small Signal Analysis. (Default = 0).

Offset - the DC offset of the signal generator (in Amps). (Default = 2.5).

Amplitude - the peak amplitude of the output current (in Amps). (Default = 1).

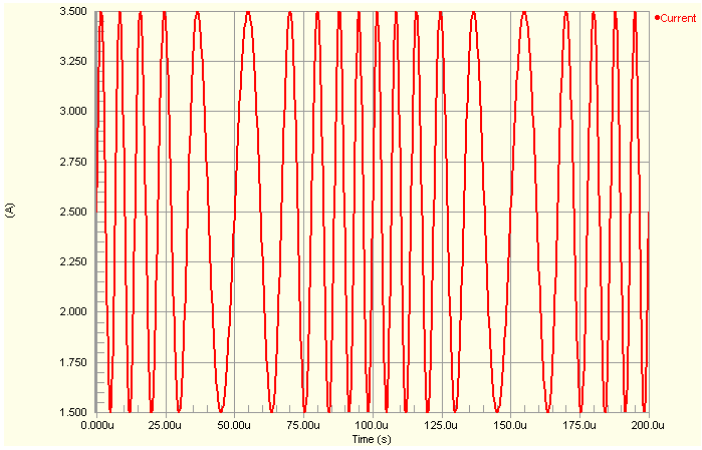
Carrier Frequency - the carrier frequency (in Hz). (Default = 100k).

Modulation Index - the modulation index (Default = 5).

Signal Frequency - the signal (message) frequency (in Hz). (Default = 10k).

Notes

The adjacent image shows an example waveform produced by an FM current source (connected to a 1Ohm load) with the parameters set to their default values:



The shape of the waveform is described by the following formula:

$$I(t) = I_O + I_A \sin[2\pi F_C t + MI \sin(2\pi F_S t)]$$

where,

t is an instance of time

I_O is the DC offset of the signal generator

I_A is the maximum amplitude of the output swing (excluding the DC offset)

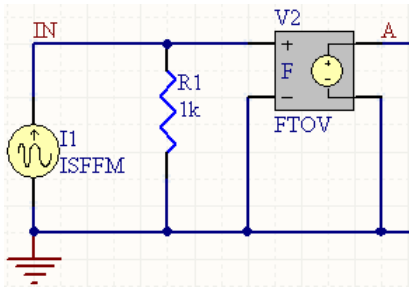
F_C is the Carrier frequency

MI is the Modulation Index and

F_S is the Signal frequency.

The simulation-ready frequency modulated sinusoidal current source component (`ISFFM`) can be found in the Simulation Sources integrated library (`\Library\Simulation\Simulation Sources.IntLib`).

Examples



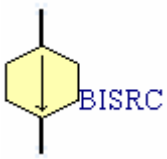
Consider the frequency modulated sinusoidal current source in the previous image, with the following characteristics:

- Pin1 (positive) is connected to net `IN`
- Pin2 (negative) is connected to net `GND`
- Designator is `I1`
- Offset = `0`
- Amplitude = `1m`
- Carrier Frequency = `10k`
- Signal Frequency = `1k`
- All other parameters for the model are left at their default values.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
I1 IN 0 DC 0 SFFM(0 1m 10k 5 1k) AC 1 0
```

Non-Linear Dependent Current Source



Model Kind

Current Source

Model Sub-Kind

Equation

SPICE Prefix

B

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 I=@EQUATION
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Equation - expression defining the source waveform.

Notes

Standard SPICE non-linear dependant current source. This source is sometimes called an Equation-defined source, as the output is defined by a user-defined equation, often referencing currents at other nodes in the circuit.

The current waveform is described by:

I = expression

where,

expression is the user-defined equation entered in the corresponding `Equation` parameter field.

The following standard functions can be used to create the expression:

ABS() - absolute value function. ABS(x) returns the value of |x|.

LN() - natural logarithm function, where LN(e) = 1.

SQRT() - square root function.

LOG() - log base 10 function.

EXP() - exponential function. EXP(x) returns the value of 'e raised to the power of x', where e is the base of the natural logarithms.

SIN() - sine function.

ASIN() - arc sine function.

ASINH() - hyperbolic arc sine function.

SINH() - hyperbolic sine function.

COS() - cosine function.

ACOS() - arc cosine function.

ACOSH() - hyperbolic arc cosine function.

COSH() - hyperbolic cosine function.

TAN() - tangent function.

ATAN() - arc tangent function.

ATANH() - hyperbolic arc tangent function.

U() - unit step function. Returns a value of 1 for arguments greater than 0 and a value of 0 for arguments less than 0.

URAMP() - unit ramp function. Integral of the unit step: for an input x, the value is 0 if x is less than 0, or if x is greater than 0, the value is x.

The following standard operators are supported:

+ - addition operator.

- - subtraction operator.

***** - multiplication operator.

/ - division operator.

^ - power operator. y^x returns the value of 'y raised to the power of x'.

unary - - unary minus operator. unary -x returns -x.

To reference in an equation the current at a node in your circuit, you must first name the node in the schematic using a Net Label. You then use the name defined in the Net field of the Net Label's properties to reference the node using the following syntax:

`I(Net)` - references the current at node `Net`

For example, if you have a node in your circuit labeled with a Net Label called `IN`, then the following would be valid entries in the `Equation` parameter field of the source:

```
I(IN)^3
COS(I(IN))
```

By default, the node is referenced to the **Spice Reference Net Name**, specified on the **Spice Options** page of the *Analyses Setup* dialog. This is `GND` by default. You can include a different reference node directly in the equation using the following syntax:

```
I(netlabel1, netlabel2)
```

For example:

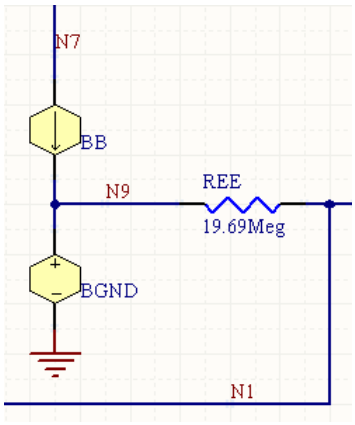
```
LN(COS(LOG(I(NetLabel1,NetLabel2)^2))) - I(NetLabel2)^I(NetLabel1)
```

If the argument of a `LOG()`, `LN()`, or `SQRT()` function becomes less than zero, the absolute value of the argument is used.

If a divisor becomes zero or the argument of log or ln becomes zero, an error will result. Other problems may occur when the argument for a function in a partial derivative enters a region where that function is undefined.

The simulation-ready non-linear dependent current source component (`BISRC`) can be found in the Simulation Sources integrated library (`\Library\Simulation\Simulation Sources.IntLib`).

Examples



Consider the non-linear dependent current source in the above image, with the following characteristics:

- Pin1 (positive) is connected to net `N7`
- Pin2 (negative) is connected to net `N9`
- Designator is `BB`
- Equation = `I(VB)*10.61E6-I(VC)*10E6+I(VE)*10E6+I(VLP)*10E6-I(VLN)*10E6.`

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
BB N7 N9 I=I(VB)*10.61E6-I(VC)*10E6+I(VE)*10E6+I(VLP)*10E6-I(VLN)*10E6
```

Piecewise Linear Current Source

Model Kind

Current Source

Model Sub-Kind

Piecewise Linear

SPICE Prefix

I

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 ?"DC MAGNITUDE"|DC @"DC MAGNITUDE"|
PWL(?MODELLOCATION/FILE=@MODELLOCATION//@"TIME-VALUE PAIRS"/) #"AC MAGNITUDE"|AC @"AC
MAGNITUDE"| @"AC PHASE"
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

DC Magnitude	-	DC offset used in an Operating Point Analysis. (Default = 0).
AC Magnitude	-	the magnitude of the source when used in an AC Small Signal Analysis. (Default = 1).
AC Phase	-	the phase of the source when used in an AC Small Signal Analysis. (Default = 0).
Time / Value Pairs	-	allows you to define the waveform by specifying a value for the current at various points in time. (Default pairings are: `0U 5A, 5U 5A, 12U 0A, 50U 5A, 60U 5A`).

Notes

Piecewise linear sources can take data from one of two sources:

- You can describe the waveform with a set of points that you enter directly into the `Time/Value Pairs` list, on the **Parameters** tab of the *Sim Model* dialog. Use the available **Add** and **Delete** buttons to define new points or remove existing ones respectively. There is no upper limit on the number of points you can define for the waveform. The time specified for each successive point must be more positive than its predecessor. If it is not the cycle will end, excluding that and all successive points.
- You can define the waveform in an ASCII text file containing an indefinite number of points. The file must be stored in the same location as the parent project file, with the extension `.PWL`. The file is referenced by entering its name (including extension) in the **Model Location** region's **In File** field, on the **Model Kind** tab of the *Sim Model* dialog.

  The following criteria must be adhered to when defining the data in the file:
- Values must be entered in pairs: a time position followed by an amplitude.
- The first character of each data line must be a plus sign (+) and each line may contain up to 255 characters.
- Values must be separated by one or more spaces or tabs.
- Values may be entered in either scientific or engineering notation.
- Comment lines may be added by making the first character of the line an asterisk (*).

  The following example illustrates the typical format for the content in a `.pwl` file:

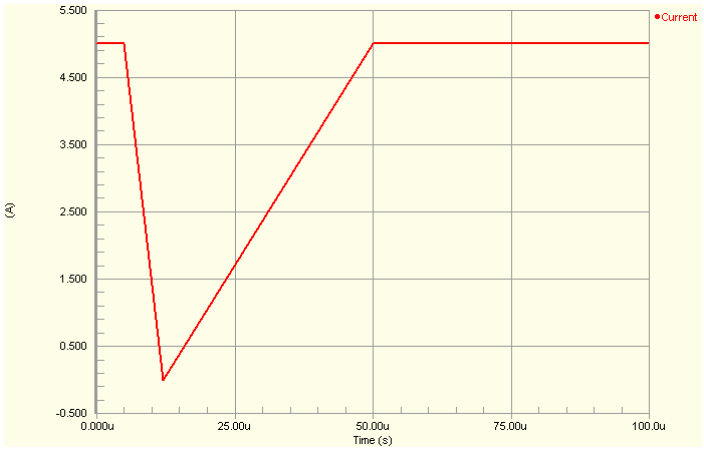
```
* Random Noise Data

+ 0.00000e-3       0.6667      0.00781e-3       0.6372      0.01563e-3       -0.1177

+ 0.02344e-3       -0.6058     0.03125e-3       0.2386      0.03906e-3       -1.1258

+ 0.04688e-3       1.6164      0.05469e-3       -0.3136     0.06250e-3       -1.0934
```

The adjacent image shows an example waveform produced by a PWL current source (connected to a 1Ohm load) with the parameters set to their default values:

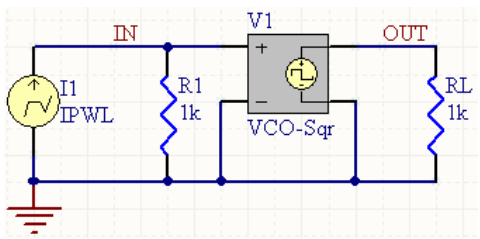


The value for the current at intermediate values of time is calculated using linear interpolation on input values.

The value of the current at time points subsequent to the last time point defined, will be the current value defined for that last time point. Similarly, if the waveform has been described starting at a time other than zero, all points in time back to zero will have that current which is defined for the first time point of the waveform.

The simulation-ready piecewise-linear current source component (`IPWL`) can be found in the Simulation Sources integrated library (`\Library\Simulation\Simulation Sources.IntLib`).

Examples



Consider the piecewise-linear current source in the above image, with the following characteristics:

- Pin1 (positive) is connected to net `GND`
- Pin2 (negative) is connected to net `IN`
- Designator is `I1`
- Time/Value Pair entries are:

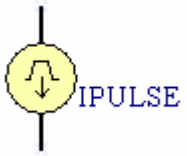
Time (s)	Current (A)
0	1e-4
2m	3e-4
4m	1e-3
6m	1.5e-3
8m	5e-4
10m	2.5e-3
12m	4e-3
14m	1e-4

- All other parameters for the model are left at their default values.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
I1 0 IN DC 0 PWL(0 1e-4 2m 3e-4 4m 1e-3 6m 1.5e-3 8m 5e-4 10m 2.5e-3 12m 4e-3 14m
+ 1e-4) AC 1 0
```

Pulse Current Source



Model Kind

Current Source

Model Sub-Kind

Pulse

SPICE Prefix

I

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 ?"DC MAGNITUDE"|DC @"DC MAGNITUDE"| PULSE(?"INITIAL VALUE"/&"INITIAL
VALUE"//0/ ?"PULSED VALUE"/&"PULSED VALUE"//5/ ?"TIME DELAY"/&"TIME DELAY"//0/ ?"RISE
TIME"/&"RISE TIME"//4U/ ?"FALL TIME"/&"FALL TIME"//1U/ ?"PULSE WIDTH"/&"PULSE WIDTH"//0/
?PERIOD/&PERIOD//5U/ &PHASE) #"AC MAGNITUDE"|AC @"AC MAGNITUDE"| @"AC PHASE"
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

DC Magnitude - DC offset used in an Operating Point Analysis. (Default = 0).

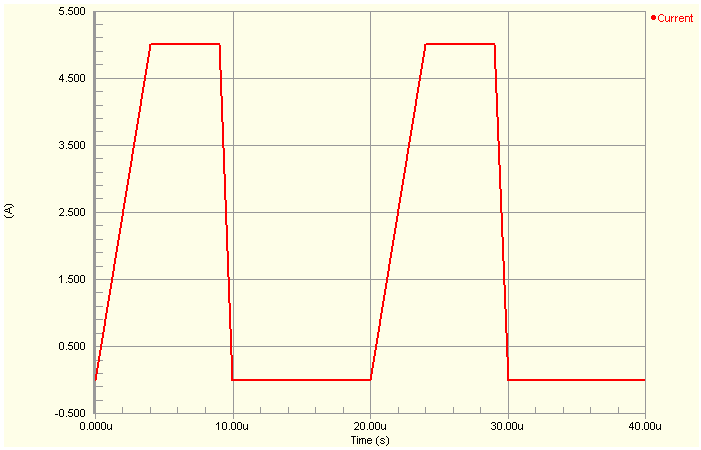
AC Magnitude - the magnitude of the source when used in an AC Small Signal Analysis. (Default = 1).

AC Phase	-	the phase of the source when used in an AC Small Signal Analysis. (Default = 0).
Initial Value	-	current amplitude at time zero (in Amps). (Default = 0).
Pulsed Value	-	maximum amplitude of the output swing (in Amps). (Default = 5).
Time Delay	-	delay before the source changes from Initial current value to Pulsed current value (in seconds).
Rise Time	-	the time it takes to rise from Initial current value to Pulsed current value (in seconds). Must be > 0. (Default = 4u).
Fall Time	-	the time it takes to fall from Pulsed current value back to the Initial current value (in seconds). Must be > 0. (Default = 1u).
Pulse Width	-	the time that the source remains at the Pulsed current amplitude (in seconds). (Default = 0).
Period		the time between the start of the first pulse and the start of the second pulse (in seconds). (Default = 5u).
Phase	-	phase shift of the waveform at time zero (in Degrees).

Notes

The adjacent image shows an example waveform produced by a periodic-pulse current source (connected to a 1Ohm load). The `Pulse Width` has been set to `5u`, the `Period` has been set to `20u`. All other parameters have been left at their default values:

The shape of the waveform is described as follows:



$I(t_0) = I_{IV}$

$I(t_{TD}) = I_{IV}$

$I(t_{TD} + t_{RT}) = I_{PV}$

$I(t_{TD} + t_{RT} + t_{PW}) = I_{PV}$

$I(t_{TD} + t_{RT} + t_{PW} + t_{FT}) = I_{IV}$

$I(t_{STOP}) = I_{IV}$

where,

t is an instance of time

I_{IV} is the initial value of the current

I_{PV} is the pulsed value of the current

t_{TD} is the Time Delay

t_{RT} is the Rise Time

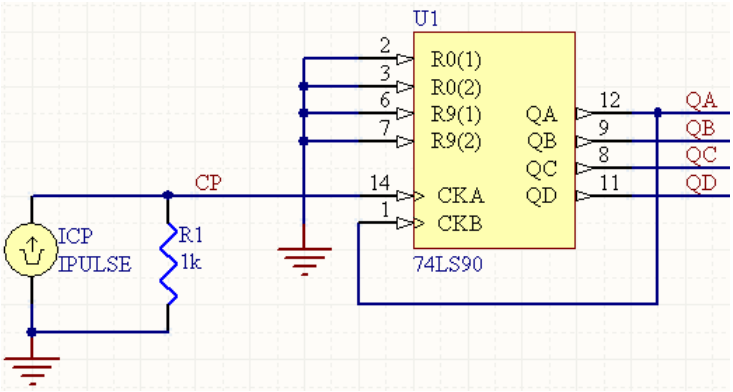
t_{PW} is the Pulse Width and

t_{FT} is the Fall Time.

The value for the current at intermediate values of time is calculated using linear interpolation.

The simulation-ready pulse current source component (`IPULSE`) can be found in the Simulation Sources integrated library (`\Library\Simulation\Simulation Sources.IntLib`).

Examples



Consider the pulse current source in the above image, with the following characteristics:

- Pin1 (positive) is connected to net `GND`
- Pin2 (negative) is connected to net `CP`
- Designator is `ICP`
- Pulsed Value = `5m`
- Time Delay = `0`
- Rise Time = `1u`
- Pulse Width = `500u`
- Period = `1000u`
- All other parameters for the model are left at their default values.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
ICP 0 CP DC 0 PULSE(0 5m 0 1u 1u 500u 1000u) AC 1 0
```

Sinusoidal Current Source

Model Kind

Current Source

Model Sub-Kind

Sinusoidal

SPICE Prefix

I

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 ?"DC MAGNITUDE"|DC @"DC MAGNITUDE"| SIN(?OFFSET/&OFFSET//0/
?AMPLITUDE/&AMPLITUDE//1/ ?FREQUENCY/&FREQUENCY//1K/ ?DELAY/&DELAY//0/ ?"DAMPING
FACTOR"/&"DAMPING FACTOR"//0/ &PHASE) #"AC MAGNITUDE"|AC @"AC MAGNITUDE"| @"AC PHASE"
```

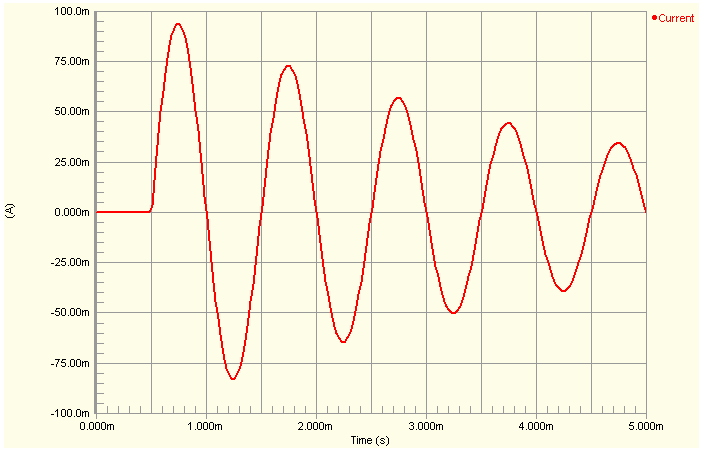
Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

DC Magnitude	-	DC offset used in an Operating Point Analysis. (Default = 0).
AC Magnitude	-	the magnitude of the source when used in an AC Small Signal Analysis. (Default = 1).
AC Phase	-	the phase of the source when used in an AC Small Signal Analysis. (Default = 0).
Offset	-	DC offset current of the signal generator (in Amps). (Default = 0).
Amplitude	-	peak amplitude of the sinusoid (in Amps). (Default = 1).
Frequency	-	frequency of the sinusoidal output current (in Hz). (Default = 1K).
Delay	-	delay time until the source current commences (in seconds). (Default = 0).
Damping Factor	-	the rate at which the sinusoid decreases/increases in amplitude (in 1/seconds). A positive value results in an exponentially decreasing amplitude; a negative value gives an increasing amplitude. A zero (0) value gives a constant amplitude sine wave. (Default = 0).
Phase	-	phase shift of the sinusoid at time zero (in Degrees). (Default = 0).

Notes

The adjacent image shows an example waveform produced by a sinusoidal current source (connected to a 1Ohm load). The `Amplitude` has been set to `100mA`, the `Delay` set to `500.0u` and the `Damping Factor` set to `250` – to illustrate a decreasing sinusoid. All other parameters have been left at their default values:



The shape of the waveform is described by the following formulae:

$I(t_0 \text{ to } t_D) = I_O$

$I(t_D \text{ to } t_{STOP}) = I_O + I_A\, e^{-(t - t_D)^{THETA}} \sin(2\pi F\,(t + t_D))$

where,

t is an instance of time

I_O is the DC offset current of the signal generator

I_A is the maximum amplitude of the output swing (excluding the DC offset)

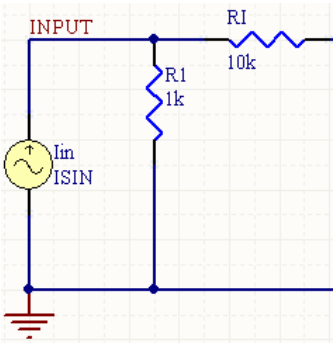
F is the Frequency

t_D is the Delay and

THETA is the Damping Factor.

The simulation-ready sinusoidal current source component (`ISIN`) can be found in the Simulation Sources integrated library (`Library\Simulation\Simulation Sources.IntLib`).

Examples



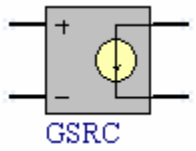
Consider the sinusoidal voltage source in the above image, with the following characteristics:

- Pin1 (positive) is connected to net `GND`
- Pin2 (negative) is connected to net `INPUT`
- Designator is `Iin`
- Amplitude = `1m`
- Frequency = `10k`
- All other parameters for the model are left at their default values.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
Iin 0 INPUT DC 0 SIN(0 1m 10K 0 0 0) AC 1 0
```

Voltage-Controlled Current Source



Model Kind

Current Source

Model Sub-Kind

Voltage-Controlled

SPICE Prefix

G

SPICE Netlist Template Format

```
@DESIGNATOR %3 %4 %1 %2 @GAIN
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Gain - transconductance of the source (in mhos).

Notes

This source produces a current at the output terminals that is a linear function of the voltage at the input terminals, dependant on the transconductance of the source.

The characteristic equation for this source is:

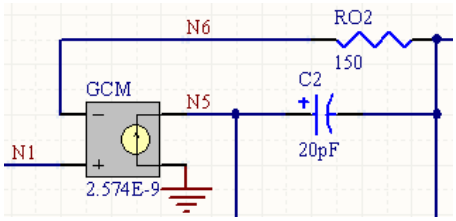
i = gv

where,

g is the transconductance.

The simulation-ready voltage controlled current source component (GSRC) can be found in the Simulation Sources integrated library (\Library\Simulation\Simulation Sources.IntLib).

Examples



Consider the voltage-controlled current source in the above image, with the following characteristics:

- Pin1 (positive controlling node) is connected to net N1
- Pin2 (negative controlling node) is connected to net N6
- Pin3 (positive output node) is connected to net GND
- Pin4 (negative output node) is connected to net N5
- Designator is GCM
- Gain = 2.574E-9.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
GCM 0 N5 N1 N6 2.574E-9
```

PSpice Support

The following general PSpice model forms are supported:

- G<name> <(+) node> <(-) node> VALUE = { <expression> }
- G<name> <(+) node> <(-) node> TABLE { <expression> } = < <input value>,<output value> >
- G<name> <(+) node> <(-) node> POLY(<value>) < <(+) controlling node> <(-) controlling node> > < <polynomial coefficient value> >

These devices do not support linked model files. The netlist format for a PSpice model in one of the above forms should be specified using the Generic Editor. In the *Sim Model* dialog, set the **Model Kind** to General and the **Model Sub-Kind** to Generic Editor.

For the circuit to be parsed correctly, ensure that the **Spice Prefix** field is set to G.

The following are examples of generic netlist template formats that could be used for these model types.

VALUE model

```
@DESIGNATOR %1 %2 VALUE = {@EXPR}
```

The value for the EXPR parameter is entered on the **Parameters** tab of the *Sim Model* dialog.

TABLE model

```
@DESIGNATOR %1 %2 TABLE {@EXPR} = @ROW1 ?ROW2|@ROW2| ?ROW3|@ROW3|
```

Values for the EXPR and ROW parameters are entered on the **Parameters** tab of the *Sim Model* dialog. Any number of ROW parameters can be defined, in the format (<input value>, <output value>).

The netlist format could be entered using the following alternative entry:

```
@DESIGNATOR %3 %4 TABLE { @EXPR } ( @TABLE )
```

Values for the EXPR and TABLE parameters are again entered on the **Parameters** tab of the *Sim Model* dialog. The value for the TABLE parameter is specified in the form:

```
(<input1>, <output1>)(<input2>, <output2>)...(<inputn>, <outputn>)
```

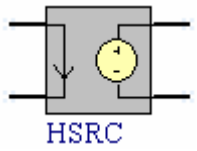
POLY model

```
@DESIGNATOR %3 %4 POLY (@dimension) (%1, %2) @coeffs
```

The values for the dimension and coeffs parameters are entered on the **Parameters** tab of the *Sim Model* dialog.

Voltage Sources

Current-Controlled Voltage Source



Model Kind

Voltage Source

Model Sub-Kind

Current-Controlled

SPICE Prefix

H

SPICE Netlist Template Format

```
V@DESIGNATOR %1 %2 0V
@DESIGNATOR %3 %4 V@DESIGNATOR @GAIN
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Gain - transresistance of the source (in Ohms).

Notes

This source produces a voltage at the output terminals that is a linear function of the current at the input terminals, dependant on the transresistance of the source.

The current-controlled voltage source actually implements two individual devices, as can be seen from the Netlist template. The first is a 0V voltage source, which acts as an ammeter, to measure the current input and then the actual current-controlled voltage source that references it. The direction of positive controlling current flow is from the positive node, through the source to the negative node of the 0V voltage source.

The characteristic equation for this source is:

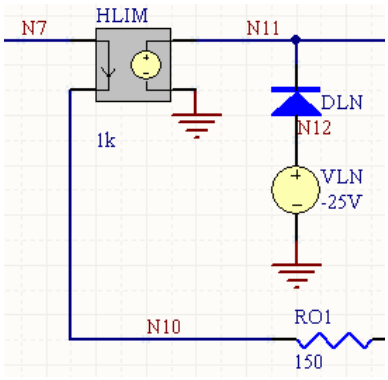
v = hi

where,

h is the transresistance.

The simulation-ready current controlled voltage source component (`HSRC`) can be found in the Simulation Sources integrated library (`\Library\Simulation\Simulation Sources.IntLib`).

Examples



Consider the current-controlled voltage source in the above image, with the following characteristics:

- Pin1 (positive controlling node) is connected to net `N7`
- Pin2 (negative controlling node) is connected to net `N10`
- Pin3 (positive output node) is connected to net `N11`
- Pin4 (negative output node) is connected to net `GND`
- Designator is `HLIM`
- Gain = `1k`.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
VHLIM N7 N10 0V
HLIM N11 0 VHLIM 1k
```

PSpice Support

The following general PSpice model form is supported:

```
H<name> <(+) node> <(-) node> POLY(<value>) <controlling V device name> < <polynomial coefficient value> >
```

This device does not support linked model files. The netlist format for a PSpice model in the above form should be specified using the Generic Editor. In the *Sim Model* dialog, set the **Model Kind** to `General` and the **Model Sub-Kind** to `Generic Editor`.

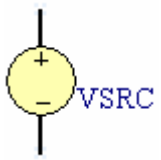
For the circuit to be parsed correctly, ensure that the **Spice Prefix** field is set to H.

The following example generic netlist template format could be used for this model type:

```
@DESIGNATOR %1 %2 POLY (@dimension) @ControlSource @coeffs
```

The values for the `dimension`, `ControlSource` and `coeffs` parameters are entered on the **Parameters** tab of the *Sim Model* dialog.

DC Voltage Source



Model Kind

Voltage Source

Model Sub-Kind

DC Source

SPICE Prefix

V

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @VALUE #"AC MAGNITUDE"|AC @"AC MAGNITUDE"| @"AC PHASE"
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Value	-	amplitude of the source voltage (in Volts).
AC Magnitude	-	the magnitude of the source when used in an AC Small Signal Analysis (typically 1V).
AC Phase	-	the phase of the source when used in an AC Small Signal Analysis.

Notes

This source produces a constant DC voltage output and is generally used to power the circuit.

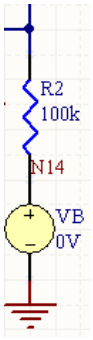
If a value for the DC source voltage is not specified, an error will occur when parsing the circuit to the Simulator.

If specifying AC criteria, the following should be observed;

- If a value for the `AC Magnitude` is entered, a value for the `AC Phase` *MUST* also be given, otherwise an error will occur when parsing the circuit.
- If a value for the `AC Magnitude` is omitted, but a value for `AC Phase` is defined, the circuit will parse to the Simulator OK, but the SPICE netlist will not contain any AC information for the source.

The simulation-ready DC voltage source component (`VSRC`) can be found in the Simulation Sources integrated library (`\Library\Simulation\Simulation Sources.IntLib`).

Examples



Consider the DC voltage source in the above image, with the following characteristics:

- Pin1 (positive) is connected to net `N14`
- Pin2 (negative) is connected to net `GND`
- Designator is `VB`
- Value = `0V`
- No AC parameters are specified.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
VB N14 0 0V
```

Exponential Voltage Source

Model Kind

Voltage Source

Model Sub-Kind

Exponential

SPICE Prefix

V

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 ?"DC MAGNITUDE"|DC @"DC MAGNITUDE"| EXP(?"INITIAL VALUE"/&"INITIAL
VALUE"//0/ ?"PULSED VALUE"/&"PULSED VALUE"//5/ ?"RISE DELAY TIME"/&"RISE DELAY TIME"//1U/
?"RISE TIME CONSTANT"/&"RISE TIME CONSTANT"//700N/ ?"FALL DELAY TIME"/&"FALL DELAY TIME"//2U/
&"FALL TIME CONSTANT") #"AC MAGNITUDE"|AC @"AC MAGNITUDE"| @"AC PHASE"
```

Parameters (definable at component level)

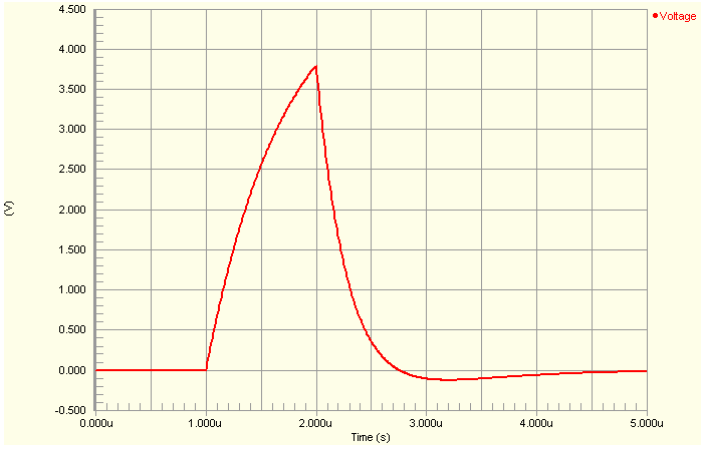
The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

DC Magnitude	-	DC offset used in an Operating Point Analysis. (Default = 0).
AC Magnitude	-	the magnitude of the source when used in an AC Small Signal Analysis. (Default = 1).
AC Phase	-	the phase of the source when used in an AC Small Signal Analysis. (Default = 0).
Initial Value	-	voltage amplitude at time zero (in Volts). (Default = 0).
Pulsed Value	-	maximum amplitude of the output swing (in Volts). (Default = 5).
Rise Delay Time	-	the point in time, from t_0, where the output begins to rise from the Initial Value to the Pulsed Value (in seconds). (Default = 1u).
Rise Time Constant	-	RC charging time constant (in seconds). (Default = 700n).
Fall Delay Time	-	the point in time, from t_0, where the output begins to fall from the Pulsed Value back to the Initial Value (in seconds). Must be >0. (Default = 2u).
Fall Time Constant	-	RC discharging time constant (in seconds). (Default = 300n).

Notes

Use this source to create a pulse voltage waveform with an exponential rising and/or falling edge.

The adjacent image shows an example waveform produced by an exponential voltage source (connected to a 1Ohm load) with the parameters set to their default values:



The shape of the waveform is described by the following formulae:

$V(t_0 \text{ to } t_{RD}) = V_{IV}$

$V(t_{RD} \text{ to } t_{FD}) = V_{IV} + (V_{PV} - V_{IV})\,(1 - e^{-(t-t_{RD})/t_{RT}})$

$V(t_{FD} \text{ to } t_{STOP}) = V_{IV} + (V_{PV} - V_{IV})\,(-e^{-(t-t_{RD})/t_{RT}}) + (V_{IV} - V_{PV})\,(1 - e^{-(t-t_{FD})/t_{FT}})$

where,

t is an instance of time

V_{IV} is the initial value of the voltage

V_{PV} is the pulsed value of the voltage

t_{RD} is the Rise Delay

t_{RT} is the Rise Time

t_{FD} is the Fall Delay and

t_{FT} is the Fall Time.

The simulation-ready exponential voltage source component (`VEXP`) can be found in the Simulation Sources integrated library (`\Library\Simulation\Simulation Sources.IntLib`).

Frequency Modulated Sinusoidal Voltage Source

Model Kind

Voltage Source

Model Sub-Kind

Single-Frequency FM

SPICE Prefix

V

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 ?"DC MAGNITUDE"|DC @"DC MAGNITUDE"| SFFM(@OFFSET @AMPLITUDE @"CARRIER FREQUENCY" @"MODULATION INDEX" @"SIGNAL FREQUENCY") #"AC MAGNITUDE"|AC @"AC MAGNITUDE"| @"AC PHASE"
```

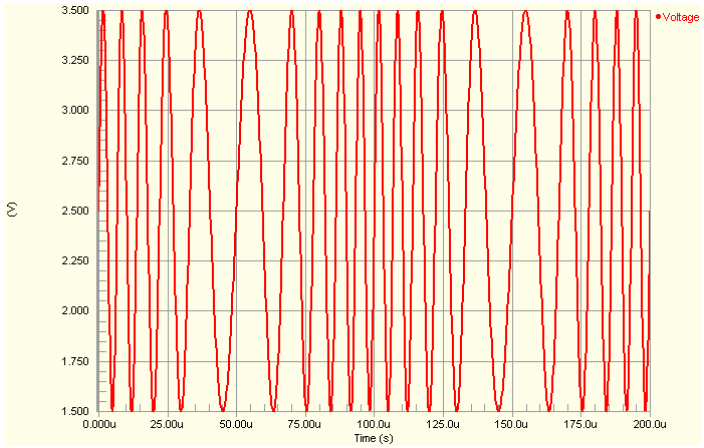
Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

DC Magnitude	-	DC offset used in an Operating Point Analysis. (Default = 0).
AC Magnitude	-	the magnitude of the source when used in an AC Small Signal Analysis. (Default = 1).
AC Phase	-	the phase of the source when used in an AC Small Signal Analysis. (Default = 0).
Offset	-	the DC offset of the signal generator (in Volts). (Default = 2.5).
Amplitude	-	the peak amplitude of the output voltage (in Volts). (Default = 1).
Carrier Frequency	-	the carrier frequency (in Hz). (Default = 100k).
Modulation Index	-	the modulation index (Default = 5).
Signal Frequency	-	the signal (message) frequency (in Hz). (Default = 10k).

Notes

The adjacent image shows an example waveform produced by an FM voltage source (connected to a 1Ohm load) with the parameters set to their default values:



The shape of the waveform is described by the following formula:

$$V(t) = V_O + V_A \sin[2\pi F_C t + MI \sin(2\pi F_S t)]$$

where,

t is an instance of time

$\mathbf{V_O}$ is the DC offset of the signal generator

$\mathbf{V_A}$ is the maximum amplitude of the output swing (excluding the DC offset)

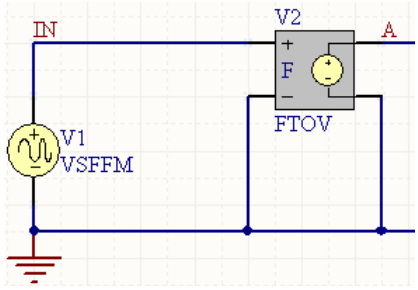
$\mathbf{F_C}$ is the Carrier frequency

MI is the Modulation Index and

$\mathbf{F_S}$ is the Signal frequency.

The simulation-ready frequency modulated sinusoidal voltage source component (`VSFFM`) can be found in the Simulation Sources integrated library (`\Library\Simulation\Simulation Sources.IntLib`).

Examples



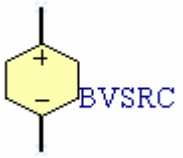
Consider the frequency modulated sinusoidal voltage source in the above image, with the following characteristics:

- Pin1 (positive) is connected to net `IN`
- Pin2 (negative) is connected to net `GND`
- Designator is `V1`
- Offset = `0`
- Carrier Frequency = `10k`
- Signal Frequency = `1k`
- All other parameters for the model are left at their default values.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
V1 IN 0 DC 0 SFFM(0 1 10k 5 1k) AC 1 0
```

Non-Linear Dependent Voltage Source



Model Kind

Voltage Source

Model Sub-Kind

Equation

SPICE Prefix

B

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 V=@EQUATION
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Equation - expression defining the source waveform.

Notes

Standard SPICE non-linear dependant voltage source. This source is sometimes called an Equation-defined source, as the output is defined by a user-defined equation, often referencing voltages at other nodes in the circuit.

The voltage waveform is described by:

V = expression

where,

expression is the user-defined equation entered in the corresponding `Equation` parameter field.

The following standard functions can be used to create the expression:

ABS() - absolute value function. ABS(x) returns the value of |x|.

LN() - natural logarithm function, where LN(e) = 1.

SQRT() - square root function.

LOG() - log base 10 function.

EXP() - exponential function. EXP(x) returns the value of 'e raised to the power of x', where e is the base of the natural logarithms.

SIN() - sine function.

ASIN() - arc sine function.

ASINH() - hyperbolic arc sine function.

SINH() - hyperbolic sine function.

COS() - cosine function.

ACOS() - arc cosine function.

ACOSH() - hyperbolic arc cosine function.

COSH() - hyperbolic cosine function.

TAN() - tangent function.

ATAN() - arc tangent function.

ATANH() - hyperbolic arc tangent function.

U() - unit step function. Returns a value of 1 for arguments greater than 0 and a value of 0 for arguments less than 0.

URAMP() - unit ramp function. Integral of the unit step: for an input x, the value is 0 if x is less than 0, or if x is greater than 0, the value is x.

The following standard operators are supported:

+ - addition operator.

- - subtraction operator.

***** - multiplication operator.

/ - division operator.

^ - power operator. y^x returns the value of 'y raised to the power of x'.

unary - - unary minus operator. unary -x returns -x.

To reference in an equation the voltage at a node in your circuit, you must first name the node in the schematic using a Net Label. You then use the name defined in the **Net** field of the Net Label's properties to reference the node using the following syntax:

`V(Net)` - references the voltage at node `Net`

For example, if you have a node in your circuit labeled with a Net Label called `IN`, then the following would be valid entries in the `Equation` parameter field of the source:

```
V(IN)^3
COS(V(IN))
```

By default, the node is referenced to the **Spice Reference Net Name**, specified on the **Spice Options** page of the *Analyses Setup* dialog. This is `GND` by default. You can include a different reference node directly in the equation using the following syntax:

```
V(netlabel1, netlabel2)
```

For example:

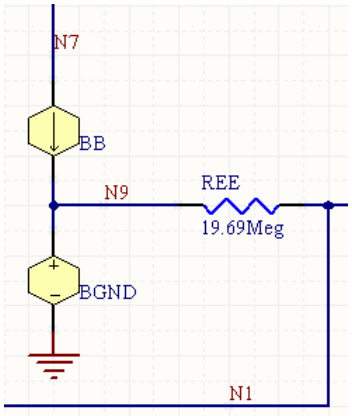
```
LN(COS(LOG(V(NetLabel1,NetLabel2)^2))) - V(NetLabel2)^V(NetLabel1)
```

If the argument of a `LOG()`, `LN()`, or `SQRT()` function becomes less than zero, the absolute value of the argument is used.

If a divisor becomes zero or the argument of log or ln becomes zero, an error will result. Other problems may occur when the argument for a function in a partial derivative enters a region where that function is undefined.

The simulation-ready non-linear dependent voltage source component (`BVSRC`) can be found in the Simulation Sources integrated library (`\Library\Simulation\Simulation Sources.IntLib`).

Examples



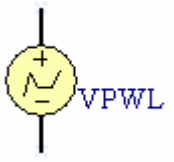
Consider the non-linear dependent voltage source in the above image, with the following characteristics:

- Pin1 (positive) is connected to net `N9`
- Pin2 (negative) is connected to net `GND`
- Designator is `BGND`
- Equation = `V(VCC)*.5+V(VEE)*.5`.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
BGND N9 0 V=V(VCC)*.5+V(VEE)*.5
```

Piecewise Linear Voltage Source



Model Kind

Voltage Source

Model Sub-Kind

Piecewise Linear

SPICE Prefix

V

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 ?"DC MAGNITUDE"|DC @"DC MAGNITUDE"|
PWL(?MODELLOCATION/FILE=@MODELLOCATION//@"TIME-VALUE PAIRS"/) #"AC MAGNITUDE"|AC @"AC
MAGNITUDE"| @"AC PHASE"
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

DC Magnitude - DC offset used in an Operating Point Analysis. (Default = 0).

AC Magnitude - the magnitude of the source when used in an AC Small Signal Analysis. (Default = 1).

AC Phase - the phase of the source when used in an AC Small Signal Analysis. (Default = 0).

Time / Value Pairs - allows you to define the waveform by specifying a value for the voltage at various points in time. (Default pairings are: `0U 5V, 5U 5V, 12U 0V, 50U 5V, 60U 5V`).

Notes

Use this source to create an arbitrary waveform as a set of voltages at various points in time.

Piecewise linear sources can take data from one of two sources:

- You can describe the waveform with a set of points that you enter directly into the `Time/Value Pairs` list, on the **Parameters** tab of the *Sim Model* dialog. Use the available **Add** and **Delete** buttons to define new points or remove existing ones respectively. There is no upper limit on the number of points you can define for the waveform. The time specified for each successive point must be more positive than its predecessor. If it is not the cycle will end, excluding that and all successive points.

- You can define the waveform in an ASCII text file containing an indefinite number of points. The file must be stored in the same location as the parent project file, with the extension `.PWL`. The file is referenced by entering its name (including extension) in the **Model Location** region's **In File** field, on the **Model Kind** tab of the *Sim Model* dialog.

The following criteria must be adhered to when defining the data in the file:

- Values must be entered in pairs: a time position followed by an amplitude.
- The first character of each data line must be a plus sign (+) and each line may contain up to 255 characters.
- Values must be separated by one or more spaces or tabs.
- Values may be entered in either scientific or engineering notation.
- Comment lines may be added by making the first character of the line an asterisk (*).

The following example illustrates the typical format for the content in a `.pwl` file:

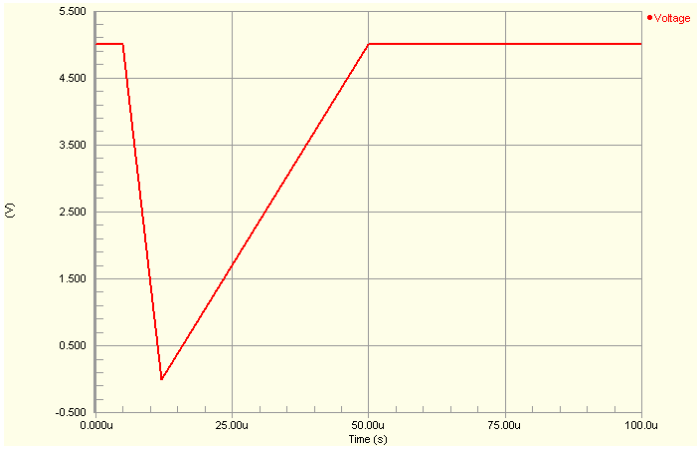
```
* Random Noise Data

+ 0.00000e-3       0.6667      0.00781e-3       0.6372      0.01563e-3       -0.1177

+ 0.02344e-3       -0.6058     0.03125e-3       0.2386      0.03906e-3       -1.1258

+ 0.04688e-3       1.6164      0.05469e-3       -0.3136     0.06250e-3       -1.0934
```

The adjacent image shows an example waveform produced by a PWL voltage source (connected to a 1Ohm load) with the parameters set to their default values:

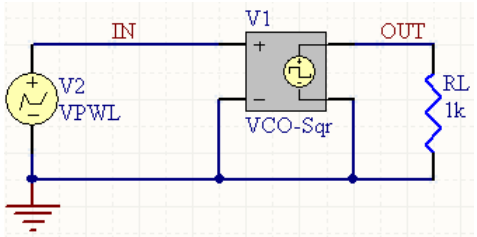


The value for the voltage at intermediate values of time is calculated using linear interpolation on input values.

The value of the voltage at time points subsequent to the last time point defined, will be the voltage value defined for that last time point. Similarly, if the waveform has been described starting at a time other than zero, all points in time back to zero will have that voltage which is defined for the first time point of the waveform.

The simulation-ready piecewise-linear voltage source component (`VPWL`) can be found in the Simulation Sources integrated library (`\Library\Simulation\Simulation Sources.IntLib`).

Examples



Consider the piecewise-linear voltage source in the above image, with the following characteristics:

- Pin1 (positive) is connected to net `IN`
- Pin2 (negative) is connected to net `GND`
- Designator is `V2`
- Time/Value Pair entries are:

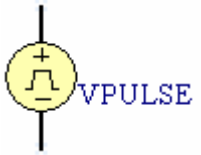
Time (s)	Voltage (V)
0	.1
2m	.3
4m	1
6m	1.5
8m	.5
10m	2.5
12m	4
14m	.1

- All other parameters for the model are left at their default values.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
V2 IN 0 DC 0 PWL(0 .1 2m .3 4m 1 6m 1.5 8m .5 10m 2.5 12m 4 14m .1) AC 1 0
```

Pulse Voltage Source



Model Kind

Voltage Source

Model Sub-Kind

Pulse

SPICE Prefix

V

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 ?"DC MAGNITUDE"|DC @"DC MAGNITUDE"| PULSE(?"INITIAL VALUE"/&"INITIAL
VALUE"//0/ ?"PULSED VALUE"/&"PULSED VALUE"//5/ ?"TIME DELAY"/&"TIME DELAY"//0/ ?"RISE
TIME"/&"RISE TIME"//4U/ ?"FALL TIME"/&"FALL TIME"//1U/ ?"PULSE WIDTH"/&"PULSE WIDTH"//0/
?PERIOD/&PERIOD//5U/ &PHASE) #"AC MAGNITUDE"|AC @"AC MAGNITUDE"| @"AC PHASE"
```

Parameters (definable at component level)

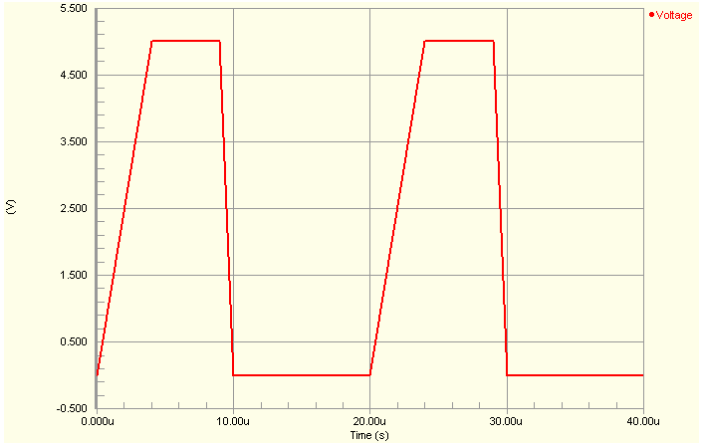
The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

DC Magnitude	-	DC offset used in an Operating Point Analysis. (Default = 0).
AC Magnitude	-	the magnitude of the source when used in an AC Small Signal Analysis. (Default = 1).
AC Phase	-	the phase of the source when used in an AC Small Signal Analysis. (Default = 0).
Initial Value	-	voltage amplitude at time zero (in Volts). (Default = 0).
Pulsed Value	-	maximum amplitude of the output swing (in Volts). (Default = 5).
Time Delay	-	delay before the source changes from Initial voltage value to Pulsed voltage value (in seconds).

Rise Time	-	the time it takes to rise from Initial voltage value to Pulsed voltage value (in seconds). Must be > 0. (Default = 4u).
Fall Time	-	the time it takes to fall from Pulsed voltage value back to the Initial voltage value (in seconds). Must be > 0. (Default = 1u).
Pulse Width	-	the time that the source remains at the Pulsed voltage amplitude (in seconds). (Default = 0).
Period	-	the time between the start of the first pulse and the start of the second pulse (in seconds). (Default = 5u).
Phase	-	phase shift of the waveform at time zero (in Degrees).

Notes

The adjacent image shows an example waveform produced by a periodic-pulse voltage source (connected to a 1Ohm load). The `Pulse Width` has been set to `5u`, the `Period` has been set to `20u`. All other parameters have been left at their default values:



The shape of the waveform is described as follows:

$V(t_0)$	=	V_{IV}
$V(t_{TD})$	=	V_{IV}
$V(t_{TD} + t_{RT})$	=	V_{PV}
$V(t_{TD} + t_{RT} + t_{PW})$	=	V_{PV}
$V(t_{TD} + t_{RT} + t_{PW} + t_{FT})$	=	V_{IV}
$V(t_{STOP})$	=	V_{IV}

where,

t is an instance of time

V_{IV} is the initial value of the voltage

V_{PV} is the pulsed value of the voltage

t_{TD} is the Time Delay

t_{RT} is the Rise Time

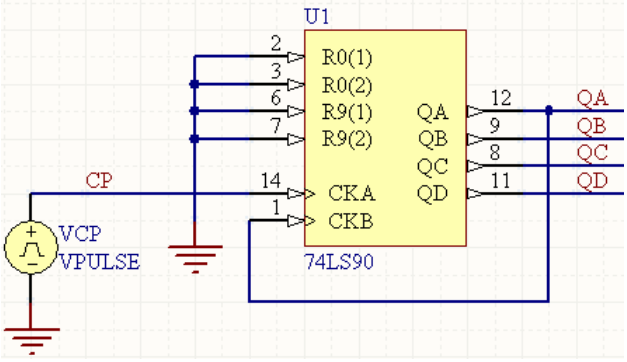
t_{PW} is the Pulse Width and

t_{FT} is the Fall Time.

The value for the voltage at intermediate values of time is calculated using linear interpolation.

The simulation-ready pulse voltage source component (`VPULSE`) can be found in the Simulation Sources integrated library (`\Library\Simulation\Simulation Sources.IntLib`).

Examples



Consider the pulse voltage source in the above image, with the following characteristics:

- Pin1 (positive) is connected to net `CP`
- Pin2 (negative) is connected to net `GND`
- Designator is `VCP`
- Time Delay = `0`
- Rise Time = `1u`
- Pulse Width = `500u`
- Period = `1000u`
- All other parameters for the model are left at their default values.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
VCP CP 0 DC 0 PULSE(0 5 0 1u 1u 500u 1000u) AC 1 0
```

Sinusoidal Voltage Source

Model Kind

Voltage Source

Model Sub-Kind

Sinusoidal

SPICE Prefix

V

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 ?"DC MAGNITUDE"|DC @"DC MAGNITUDE"| SIN(?OFFSET/&OFFSET//0/
?AMPLITUDE/&AMPLITUDE//1/ ?FREQUENCY/&FREQUENCY//1K/ ?DELAY/&DELAY//0/ ?"DAMPING
FACTOR"/&"DAMPING FACTOR"//0/ &PHASE) #"AC MAGNITUDE"|AC @"AC MAGNITUDE"| @"AC PHASE"
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

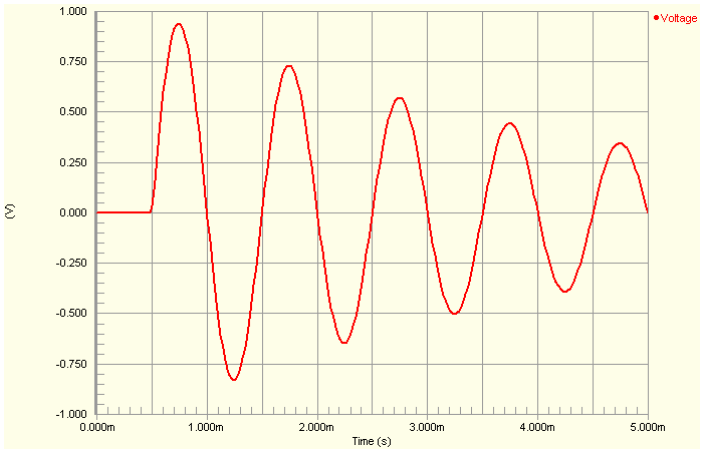
DC Magnitude - DC offset used in an Operating Point Analysis. (Default = 0).

AC Magnitude - the magnitude of the source when used in an AC Small Signal Analysis. (Default = 1).

AC Phase	-	the phase of the source when used in an AC Small Signal Analysis. (Default = 0).
Offset	-	DC offset voltage of the signal generator (in Volts). (Default = 0).
Amplitude	-	peak amplitude of the sinusoid (in Volts). (Default = 1).
Frequency	-	frequency of the sinusoidal output voltage (in Hz). (Default = 1K).
Delay	-	delay time until the source voltage commences (in seconds). (Default = 0).
Damping Factor	-	the rate at which the sinusoid decreases/increases in amplitude (in 1/seconds). A positive value results in an exponentially decreasing amplitude; a negative value gives an increasing amplitude. A zero (0) value gives a constant amplitude sine wave. (Default = 0).
Phase	-	phase shift of the sinusoid at time zero (in Degrees). (Default = 0).

Notes

The adjacent image shows an example waveform produced by a sinusoidal voltage source (connected to a 1Ohm load). The `Delay` has been set to `500.0u` and the `Damping Factor` set to `250` – to illustrate a decreasing sinusoid. All other parameters have been left at their default values:



The shape of the waveform is described by the following formulae:

$V(t_0 \text{ to } t_D) = V_O$

$V(t_D \text{ to } t_{STOP}) = V_O + VA\, e^{-(t - t_D)^{THETA}} \sin(2\pi F (t + t_D))$

where,

t is an instance of time

V_O is the DC offset voltage of the signal generator

V_A is the maximum amplitude of the output swing (excluding the DC offset)

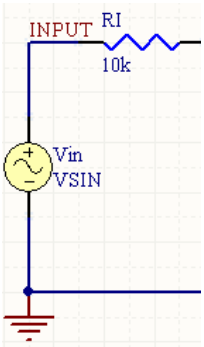
F is the Frequency

t_D is the Delay and

THETA is the Damping Factor.

The simulation-ready sinusoidal voltage source component (`VSIN`) can be found in the Simulation Sources integrated library (`\Library\Simulation\Simulation Sources.IntLib`).

Examples



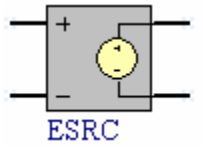
Consider the sinusoidal voltage source in the above image, with the following characteristics:

- Pin1 (positive) is connected to net `INPUT`
- Pin2 (negative) is connected to net `GND`
- Designator is `Vin`
- Frequency = `10k`
- All other parameters for the model are left at their default values.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
Vin INPUT 0 DC 0 SIN(0 1 10k 0 0) AC 1 0
```

Voltage-Controlled Voltage Source



Model Kind

Voltage Source

Model Sub-Kind

Voltage-Controlled

SPICE Prefix

E

SPICE Netlist Template Format

```
@DESIGNATOR %3 %4 %1 %2 @GAIN
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Gain - voltage gain of the source (in Volts).

Notes

This source produces a voltage at the output terminals that is a linear function of the voltage at the input terminals, dependant on the gain of the source.

The characteristic equation for this source is:

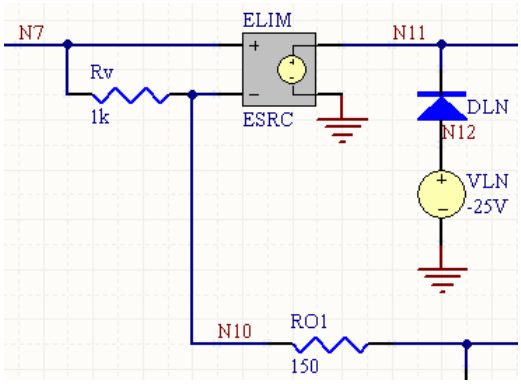
v = ev

where,

e is the voltage gain.

The simulation-ready voltage controlled voltage source component (ESRC) can be found in the Simulation Sources integrated library (\Library\Simulation\Simulation Sources.IntLib).

Examples



Consider the voltage-controlled voltage source in the above image, with the following characteristics:

- Pin1 (positive controlling node) is connected to net N7
- Pin2 (negative controlling node) is connected to net N10
- Pin3 (positive output node) is connected to net N11
- Pin4 (negative output node) is connected to net GND
- Designator is ELIM
- Gain = 1.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
ELIM N11 0 N7 N10 1
```

PSpice Support

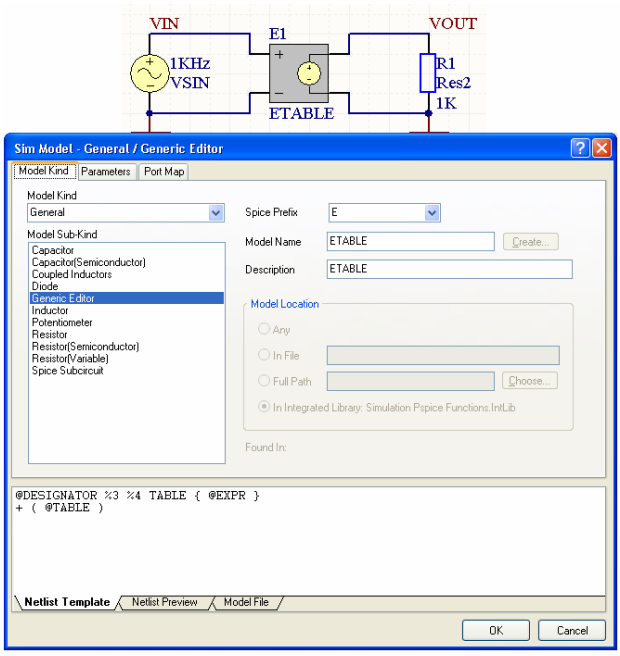
The following general PSpice model forms are supported:

- E<name> <(+) node> <(-) node> VALUE = { <expression> }
- E<name> <(+) node> <(-) node> TABLE { <expression> } = < <input value>,<output value> >
- E<name> <(+) node> <(-) node> POLY(<value>) < <(+) controlling node> <(-) controlling node> > < <polynomial coefficient value> >

These devices do not support linked model files. The netlist format for a PSpice model in one of the above forms should be specified using the Generic Editor. In the *Sim Model* dialog, set the **Model Kind** to General and the **Model Sub-Kind** to Generic Editor.

For the circuit to be parsed correctly, ensure that the **Spice Prefix** field is set to E.

The following are examples of generic netlist template formats that could be used for these model types.



VALUE model

```
@DESIGNATOR %1 %2 VALUE = {@EXPR}
```

The value for the EXPR parameter is entered on the **Parameters** tab of the *Sim Model* dialog.

TABLE model

```
@DESIGNATOR %1 %2 TABLE {@EXPR} = @ROW1 ?ROW2|@ROW2| ?ROW3|@ROW3|
```

Values for the EXPR and ROW parameters are entered on the **Parameters** tab of the *Sim Model* dialog. Any number of ROW parameters can be defined, in the format (<input value>, <output value>).

The netlist format could be entered using the following alternative entry:

```
@DESIGNATOR %3 %4 TABLE { @EXPR } ( @TABLE )
```

Values for the EXPR and TABLE parameters are again entered on the **Parameters** tab of the *Sim Model* dialog. The value for the TABLE parameter is specified in the form:

```
(<input1>, <output1>)(<input2>, <output2>)...(<inputn>, <outputn>)
```

POLY model

```
@DESIGNATOR %3 %4 POLY (@dimension) (%1, %2) @coeffs
```

The values for the dimension and coeffs parameters are entered on the **Parameters** tab of the *Sim Model* dialog.

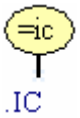
For an example of using a PSpice-compatible expression-based voltage-controlled voltage source in a simulation, refer to the example project EVALUE.PrjPCB, which can be found in the \Examples\Circuit Simulation\PSpice Examples\VCVS - Value folder of the installation.

For an example of using a PSpice-compatible lookup table-based voltage-controlled voltage source in a simulation, refer to the example project TABLE.PrjPCB, which can be found in the \Examples\Circuit Simulation\PSpice Examples\VCVS - Table folder of the installation.

For an example of using a PSpice-compatible polynomial-based voltage-controlled voltage source in a simulation, refer to the example project EPOLY.PrjPCB, which can be found in the \Examples\Circuit Simulation\PSpice Examples\Vcvs - poly folder of the installation.

Initial Conditions

Initial Condition



Model Kind

Initial Condition

Model Sub-Kind

Set Initial Condition

SPICE Prefix

None

SPICE Netlist Template Format

```
.IC V(%1)=@"INITIAL VOLTAGE"
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Initial Voltage - amplitude of the node voltage (in Volts).

Notes

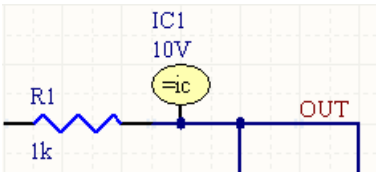
The Initial Condition device is used for setting transient initial conditions. The use of the device depends upon the setting of the **Use Initial Conditions** option, on the **Transient/Fourier Analysis Setup** page of the *Analyses Setup* dialog:

- When the **Use Initial Conditions** option is enabled, an operating point analysis is not performed. Instead, the node voltages specified by .IC devices are used to compute the capacitor, diode, BJT, JFET and MOSFET initial conditions. Since no operating point analysis is performed prior to the transient analysis, you should ensure that all appropriate DC source voltages are specified, if they are to be used to compute device initial conditions.

- When the **Use Initial Conditions** option is disabled, an operating point analysis is performed prior to the transient analysis and the node voltage is held at the value specified by the .IC device. During the subsequent transient analysis, this restraint is removed. This is the preferred method, since it allows the SPICE engine to compute a consistent DC solution.

If an Initial Condition parameter is specified for a particular device, it overrides (takes precedence over) the value defined by a .IC device.

Examples



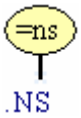
Consider the Initial Condition device in the above image, with the following characteristics:

- The pin of the device is connected to net `OUT`
- Designator is `IC1`
- Initial Voltage = `10V`.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
.IC V(OUT)=10V
```

Nodeset



Model Kind

Initial Condition

Model Sub-Kind

Initial Node Voltage Guess

SPICE Prefix

None

SPICE Netlist Template Format

```
.NODESET V(%1)=@"INITIAL VOLTAGE"
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

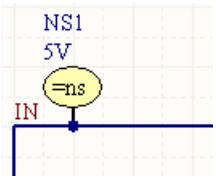
Initial Voltage - amplitude of the node voltage (in Volts).

Notes

The Nodeset device is used to specify the starting voltage for a node in the circuit during a preliminary pass of the operating point analysis. After this initial pass, the restriction is released and the iterations continue to the true bias solution.

This device is not usually necessary in order to achieve convergence in most circuits. However, it may be a useful aid when performing simulations on bistable or astable circuits.

Examples



Consider the Nodeset device in the above image, with the following characteristics:

- The pin of the device is connected to net `IN`
- Designator is `NS1`
- Initial Voltage = `5V`.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
.NODESET V(IN)=5V
```

XSpice models

These are predefined analog device code models that are built-in to XSpice. Code models allow the specification of complex, non-ideal device characteristics, without the need to develop long-winded sub-circuit definitions that can adversely affect Simulator speed performance.

The supplied models cover the following functions, which are available for operation with either single-ended or differential voltage or current I/O:

Capacitance Meter

- *Single-Ended I/O*
- *Differential I/O*

Controlled Limiter

- *Single-Ended I/O*
- *Differential I/O*

Controlled One-Shot

- *Single-Ended I/O*
- *Differential I/O*

Current Limiter

- *Single-Ended I/O*
- *Differential I/O*

Differentiator

- *Single-Ended I/O*
- *Differential I/O*

Divider

- *Single-Ended I/O*
- *Differential I/O*

Gain

- *Single-Ended I/O*
- *Differential I/O*

Hysteresis

- *Single-Ended I/O*
- *Differential I/O*

Inductance Meter

- *Single-Ended I/O*
- *Differential I/O*

Integrator

- *Single-Ended I/O*
- *Differential I/O*

Limiter

- *Single-Ended I/O*
- *Differential I/O*

Multiplier

- *Single-Ended I/O*
- *Differential I/O*

PWL Controlled Source

- *Single-Ended I/O*
- *Differential I/O*

S-Domain Transfer Function

- *Single-Ended I/O*
- *Differential I/O*

Slew Rate

- *Single-Ended I/O*
- *Differential I/O*

Summer

- *Single-Ended I/O*
- *Differential I/O*

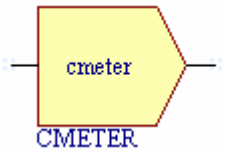
Notes

With the exception of the Multiplier and Summer functions (which are sub-circuit based variations of the models available in XSpice), the SPICE prefix for theses models is A.

All of the XSpice analog models can be found in the Simulation Special Function integrated library (`\Library\Simulation\Simulation Special Function.IntLib`). For more detailed information regarding XSpice, consult the XSpice User Manual.

Capacitance Meter

Capacitance Meter (Single-Ended I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

CMETER

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @"DESIGNATOR"CMETER
.MODEL @"DESIGNATOR"CMETER cmeter (?gain|gain=@gain|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

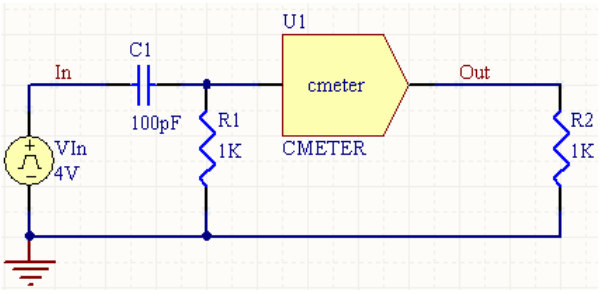
Gain - gain (default = 1).

Notes

This is a sensing device which is attached to a node in the circuit and produces as an output, a scaled value equal to the total capacitance seen on its input, multiplied by the value assigned to the `Gain` parameter. This model is useful as a building block for other models, which require to sense a capacitance value and adjust their behavior with respect to it.

The input signal can be either a single-ended current or single-ended voltage signal.

Examples



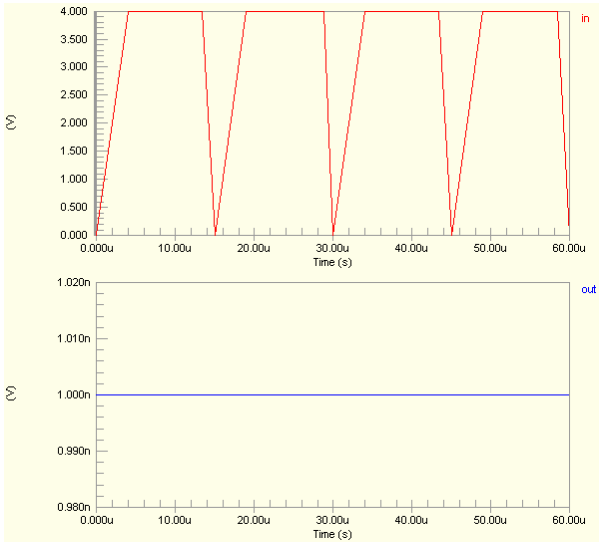
Consider the capacitance meter in the above image, with the following characteristics:

- Pin1 (input) is connected to net `NetC1_2`
- Pin2 (output) is connected to net `Out`
- Designator is `U1`
- Gain = `10`.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU1 NetC1_2 OUT AU1CMETER
.MODEL AU1CMETER cmeter (gain=10)
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

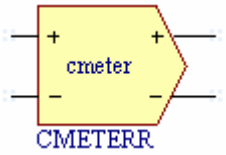


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `60.00u`
- **Transient Step Time** – set to `10.00u`
- **Transient Max Step Time** – set to `10.00u`

With the exception of the `Pulsed Value` parameter (set to `4V`) and the `Period` parameter (set to `15us`), all other parameters for the Pulse Voltage Source have been left at their defaults.

Capacitance Meter (Differential I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

CMETER

SPICE Netlist Template Format

```
@DESIGNATOR %%vd(%1,%2) %%vd(%3,%4) @"DESIGNATOR"CMETER
.MODEL @"DESIGNATOR"CMETER cmeter (?gain|gain=@gain|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

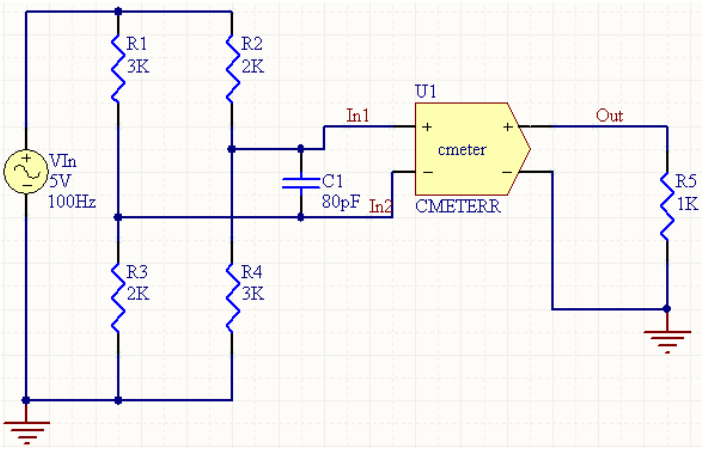
Gain - gain (default = 1).

Notes

This is a sensing device which is attached to a node in the circuit and produces as an output, a scaled value equal to the total capacitance seen on its input, multiplied by the value assigned to the `Gain` parameter. This model is useful as a building block for other models, which require to sense a capacitance value and adjust their behavior with respect to it.

The input signal can be either a differential current or differential voltage signal.

Examples



Consider the capacitance meter in the above image, with the following characteristics:

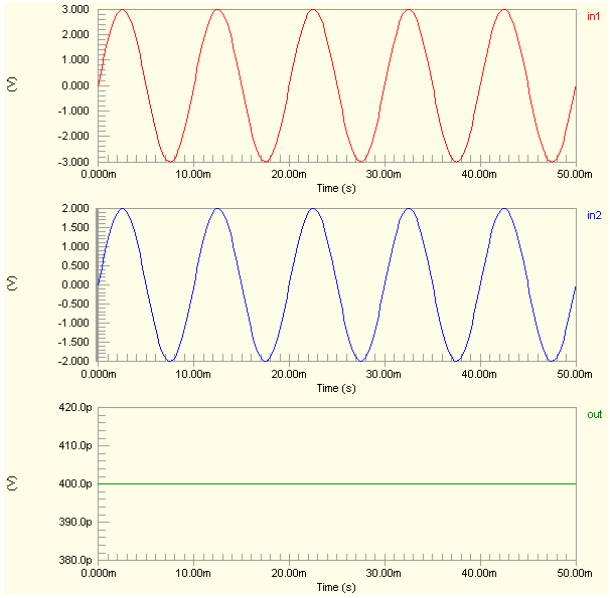
- Pin1 (positive input) is connected to net `In1`
- Pin2 (negative input) is connected to net `In2`
- Pin3 (positive output) is connected to net `Out`
- Pin4 (negative output) is connected to net `GND`

- Designator is `U1`
- Gain = `5`.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU1 %vd(IN1,IN2) %vd(OUT,0) AU1CMETER
.MODEL AU1CMETER cmeter (gain=5)
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

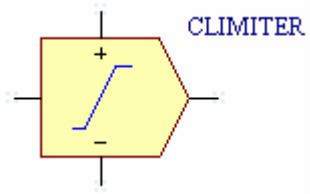


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Controlled Limiter

Controlled Limiter (Single-Ended I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

CLIMIT

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @"DESIGNATOR"CLIMIT
.MODEL @"DESIGNATOR"CLIMIT climit (?in_offset|in_offset=@in_offset| ?gain|gain=@gain|
?upper_delta|upper_delta=@upper_delta| ?lower_delta|lower_delta=@lower_delta|
?limit_range|limit_range=@limit_range| ?fraction|fraction=@fraction|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

In_Offset - input offset (Default = 0).

Gain - gain (default = 1).

Upper_Delta - output upper delta (Default = 0).

Lower_Delta - output lower delta (Default = 0).

Limit_Range - upper and lower smoothing range (Default = 1.0e-6).

Fraction - used to control whether the limit range is specified as a fractional (TRUE) or absolute (FALSE) value. (Default = FALSE).

Notes

This model is similar in function to the Gain function. However, the output is restricted to the range specified by the output lower and upper limits (`cntl_lower` and `cntl_upper` pins of the device). The input signal can be either a single-ended current or single-ended voltage signal.

The `Limit_Range` is the value below the `cntl_upper` limit and above the `cntl_lower` limit at which smoothing of the output begins. A minimum positive value of current/voltage must exist between the `cntl_upper` and `cntl_lower` inputs, at all times. The `Limit_Range` therefore represents the delta, with respect to the output level, at which smoothing occurs. For example, for an input `Gain` of 2, `Limit_Range` of 0.1V and output limits of 1V (on pin `cntl_upper`) and -1V (on pin `cntl_lower`), the output will begin to smooth out at +/-0.9 V.

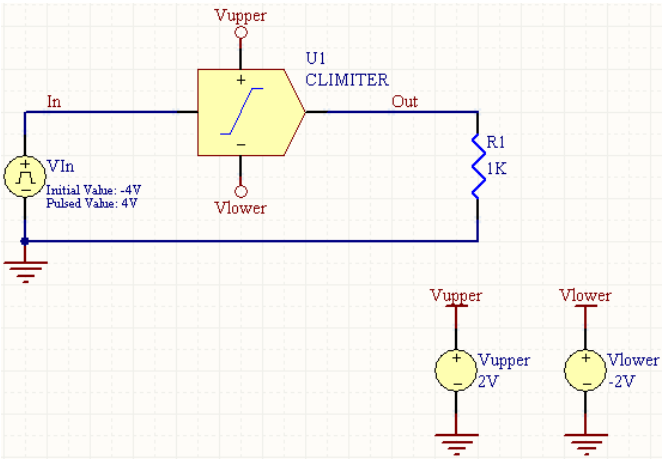
The input values arriving at the `cntl_upper` and `cntl_lower` pins of the device are tested to verify that they are far enough apart to guarantee a linear range between them. The range is calculated as:

(cntl_upper - Upper_Delta - Limit_Range) - (cntl_lower + Lower_Delta + Limit_Range)

and must be greater than or equal to zero.

When the `Limit_Range` is specified as a fractional value (`Fraction` parameter set to TRUE), it is expressed as the calculated fraction of the difference between `cntl_upper` and `cntl_lower`.

Examples



Consider the controlled limiter in the above image, with the following characteristics:

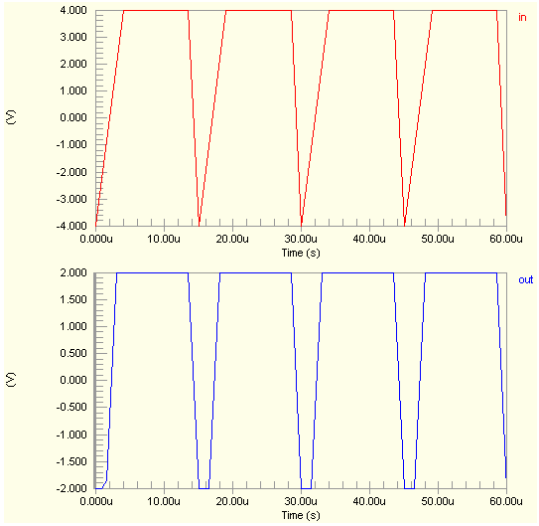
- Pin1 (input) is connected to net `In`

- Pin2 (cntl_upper) is connected to net `Vupper`
- Pin3 (cntl_lower) is connected to net `Vlower`
- Pin4 (output) is connected to net `Out`
- Designator is `U1`
- Gain = `2`
- Limit_Range = `0.1`
- All other model parameters are left at their inherent defaults.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU1 IN VUPPER VLOWER OUT AU1CLIMIT
.MODEL AU1CLIMIT climit (in_offset=0 gain=2 upper_delta=0 lower_delta=0
+ limit_range=0.1 fraction=FALSE)
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

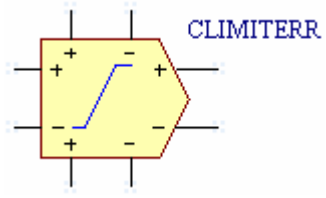


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `60.00u`
- **Transient Step Time** – set to `2.000u`
- **Transient Max Step Time** – set to `2.000u`

With the exception of the `Initial Value` parameter (set to `-4V`), the `Pulsed Value` parameter (set to `4V`) and the `Period` parameter (set to `15us`), all other parameters for the Pulse Voltage Source have been left at their defaults.

Controlled Limiter (Differential I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

CLIMIT

SPICE Netlist Template Format

```
@DESIGNATOR %%vd(%1,%2) %%vd(%3,%4) %%vd(%5,%6) %%vd(%7,%8) @"DESIGNATOR"CLIMIT
.MODEL @"DESIGNATOR"CLIMIT climit (?in_offset|in_offset=@in_offset| ?gain|gain=@gain|
?upper_delta|upper_delta=@upper_delta| ?lower_delta|lower_delta=@lower_delta|
?limit_range|limit_range=@limit_range| ?fraction|fraction=@fraction|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

In_Offset - input offset (Default = 0).

Gain - gain (default = 1).

Upper_Delta - output upper delta (Default = 0).

Lower_Delta - output lower delta (Default = 0).

Limit_Range - upper and lower smoothing range (Default = 1.0e-6).

Fraction - used to control whether the limit range is specified as a fractional (TRUE) or absolute (FALSE) value. (Default = FALSE).

Notes

This model is similar in function to the Gain function. However, the output is restricted to the range specified by the output lower and upper limits, which are defined as follows:

Upper limit = value at device pin `cntl_upper_p` - value at device pin `cntl_upper_n`

Lower limit = value at device pin `cntl_lower_p` - value at device pin `cntl_lower_n`

The input signal can be either a differential current or differential voltage signal.

The `Limit_Range` is the value below the `cntl_upper` limit and above the `cntl_lower` limit at which smoothing of the output begins. A minimum positive value of current/voltage must exist between the `cntl_upper` and `cntl_lower` inputs, at all times. The `Limit_Range` therefore represents the delta, with respect to the output level, at which smoothing occurs. For example, for an input `Gain` of 2, `Limit_Range` of 0.1V and output limits of 1V (on pin `cntl_upper`) and -1V (on pin `cntl_lower`), the output will begin to smooth out at +/-0.9 V.

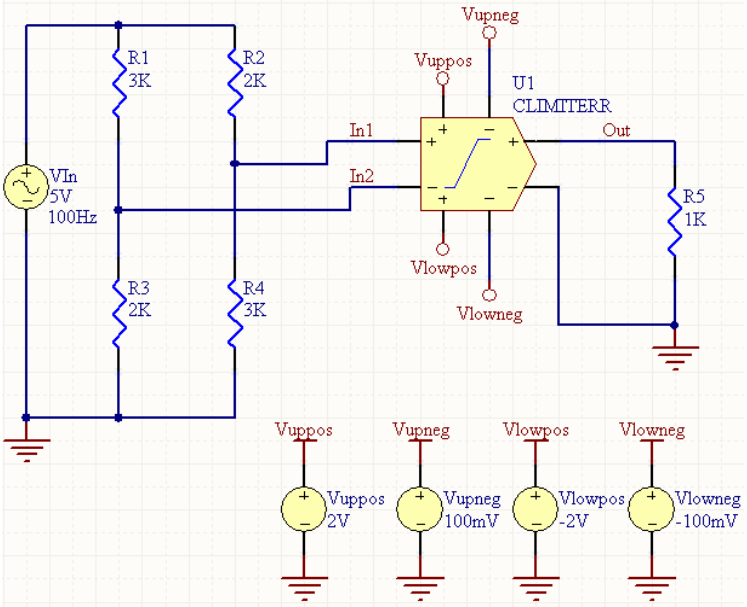
The input values arriving at the `cntl_upper` and `cntl_lower` pins of the device are tested to verify that they are far enough apart to guarantee a linear range between them. The range is calculated as:

(cntl_upper - Upper_Delta - Limit_Range) - (cntl_lower + Lower_Delta + Limit_Range)

and must be greater than or equal to zero.

When the `Limit_Range` is specified as a fractional value (`Fraction` parameter set to TRUE), it is expressed as the calculated fraction of the difference between `cntl_upper` and `cntl_lower`.

Examples



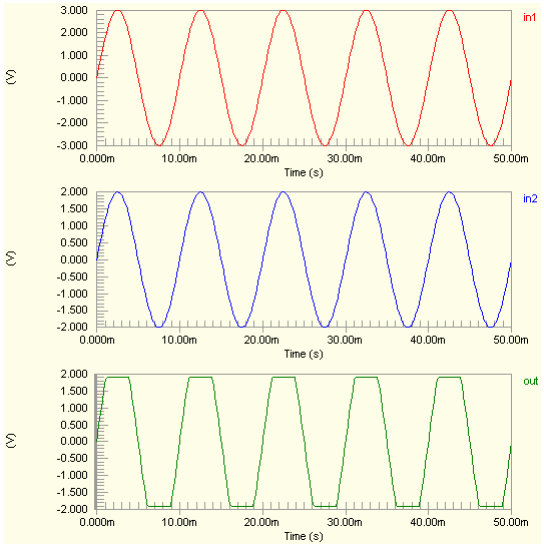
Consider the controlled limiter in the previous image, with the following characteristics:

- Pin1 (positive input) is connected to net `In1`
- Pin2 (negative input) is connected to net `In2`
- Pin3 (positive cntl_upper) is connected to net `Vuppos`
- Pin4 (negative cntl_upper) is connected to net `Vupneg`
- Pin5 (positive cntl_lower) is connected to net `Vlowpos`
- Pin6 (negative cntl_lower) is connected to net `Vlowneg`
- Pin7 (positive output) is connected to net `Out`
- Pin8 (negative output) is connected to net `GND`
- Designator is `U1`
- Gain = `3`
- All other model parameters are left at their inherent defaults.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU1 %vd(IN1,IN2) %vd(VUPPOS,VUPNEG) %vd(VLOWPOS,VLOWNEG) %vd(OUT,0) AU1CLIMIT
.MODEL AU1CLIMIT climit ( gain=3    )
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

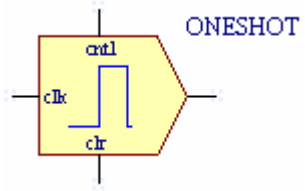


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Controlled One-Shot

Controlled One-Shot (Single-Ended I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

ONESHOT

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @"DESIGNATOR"ONESHOT
.MODEL @"DESIGNATOR"ONESHOT oneshot (?cntl_array|cntl_array=[@cntl_array]|
?pw_array|pw_array=[@pw_array]| ?clk_trig|clk_trig=@clk_trig|
?pos_edge_trig|pos_edge_trig=@pos_edge_trig| ?out_low|out_low=@out_low|
?out_high|out_high=@out_high| ?rise_time|rise_time=@rise_time|
?rise_delay|rise_delay=@rise_delay| ?fall_delay|fall_delay=@fall_delay|
?fall_time|fall_time=@fall_time| )
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Clk_Trig - clock trigger value (Default = 0.5).

cntl_array - control array (Default = 0).

Fall_Delay - delay between receiving a valid trigger level and the output starting to fall from high value to low value (Default = 1.0e-9).

Fall_Time - output fall time (Default = 1.0e-9).

Out_High - output high value (Default = 1).

Out_Low - output low value (Default = 0).

Pos_Edge_Trig - positive (TRUE)/negative (FALSE) edge trigger switch. (Default = TRUE).

Pw_Array - pulse width array. This value must be greater than or equal to zero. (Default = 1.0e-6).

Rise_Delay - delay between receiving a valid trigger level and the output starting to rise from low value to high value (Default = 1.0e-9).

Rise_Time - output rise time (Default = 1.0e-9).

Notes

This model is used to output a single pulse, the width of which is determined by a user-defined piece-wise linear waveform and a controlling input.

The `cntl_array` parameter values are input coordinate points (progressively increasing), while the `Pw_Array` parameter values represent the corresponding pulse widths at those points. You could think of the function as being analogous to a look-up table, where the input signal (`ctrl` pin of the device) amplitude is mapped to the corresponding input value in the `cntl_array` and then the `Pw_Array` value that this is paired with, is used as the width for the output pulse signal.

The amplitude of the pulse is determined by the values assigned to the `Out_Low` and `Out_High` parameters.

The output pulse is controlled by means of the `clk` input. When this input reaches the level assigned to the `Clk_Trig` parameter, the pulse is triggered, on either the rising or falling edge of the clock, in accordance with the setting of the `Pos_Edge_Trig` parameter.

Upon triggering, the output reaches its high value after time:

```
Rise_Delay + Rise_Time
```

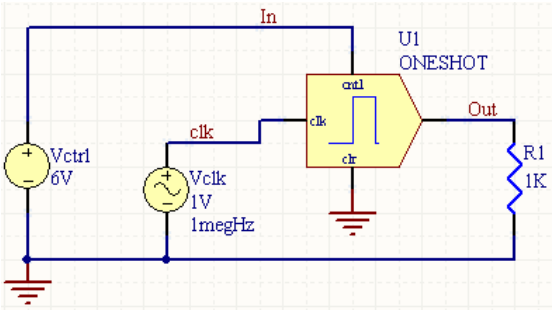
and its initial value again after time:

```
Pulse Width + Fall_Delay + Fall_Time
```

The `clr` input to the device is used to reset the state of the function so that it is possible to retrigger and thus obtain another pulse. The `clr` signal must be higher than `Clk_Trig` to achieve this.

The input signal can be either a single-ended current or single-ended voltage signal.

Examples



Consider the One-Shot function in the previous image, with the following characteristics:

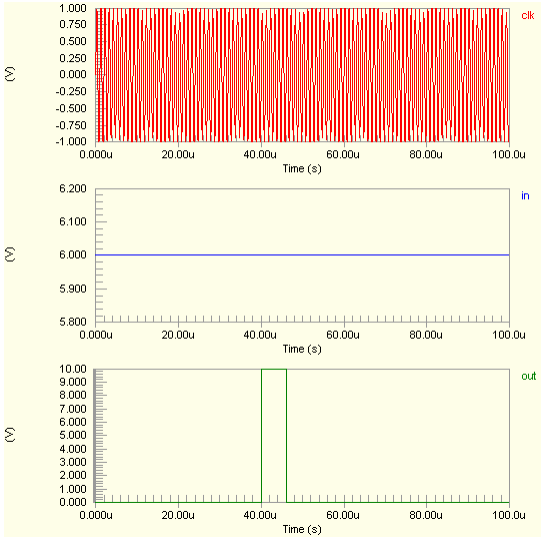
- Pin1 (clk) is connected to net `clk`

- Pin2 (ctrl) is connected to net `In1`
- Pin3 (clr) is connected to net `GND`
- Pin4 (output) is connected to net `Out`
- Designator is `U1`
- cntl_array = `1 2 3 4 5 6 7 8 9 10 11`
- Pw_Array = `1u 2u 3u 4u 5u 6u 7u 8u 9u 10u 11u`
- Clk_Trig = `0.5`
- Out_High = `10`
- Out_Low = `0`
- Pos_Edge_Trig = `TRUE`
- Rise_Delay = `40u`.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU1 CLK IN1 0 OUT AU1ONESHOT
.MODEL AU1ONESHOT oneshot (cntl_array=[1 2 3 4 5 6 7 8 9 10 11] pw_array=[1u 2u
+ 3u 4u 5u 6u 7u 8u 9u 10u 11u] clk_trig=0.5 pos_edge_trig=TRUE out_low=0
+ out_high=10  rise_delay=40u   )
```

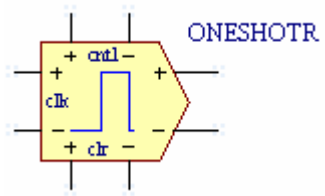
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `100.0u`
- **Transient Step Time** – set to `20.00n`
- **Transient Max Step Time** – set to `20.00n`.

Controlled One-Shot (Differential I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

ONESHOT

SPICE Netlist Template Format

```
@DESIGNATOR %%vd(%1,%2) %%vd(%3,%4) %%vd(%5,%6) %%vd(%7,%8) @"DESIGNATOR"ONESHOT
.MODEL @"DESIGNATOR"ONESHOT oneshot (?cntl_array|cntl_array=[@cntl_array]|
?pw_array|pw_array=[@pw_array]| ?clk_trig|clk_trig=@clk_trig|
?pos_edge_trig|pos_edge_trig=@pos_edge_trig| ?out_low|out_low=@out_low|
?out_high|out_high=@out_high| ?rise_time|rise_time=@rise_time|
?rise_delay|rise_delay=@rise_delay| ?fall_delay|fall_delay=@fall_delay|
?fall_time|fall_time=@fall_time| )
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Clk_Trig - clock trigger value (Default = 0.5).

cntl_array - control array (Default = 0).

Fall_Delay - delay between receiving a valid trigger level and the output starting to fall from high value to low value (Default = 1.0e-9).

Fall_Time - output fall time (Default = 1.0e-9).

Out_High - output high value (Default = 1).

Out_Low - output low value (Default = 0).

Pos_Edge_Trig - positive (TRUE)/negative (FALSE) edge trigger switch. (Default = TRUE).

Pw_Array - pulse width array. This value must be greater than or equal to zero. (Default = 1.0e-6).

Rise_Delay - delay between receiving a valid trigger level and the output starting to rise from low value to high value (Default = 1.0e-9).

Rise_Time - output rise time (Default = 1.0e-9).

Notes

This model is used to output a single pulse, the width of which is determined by a user-defined piece-wise linear waveform and a controlling input.

The `cntl_array` parameter values are input coordinate points (progressively increasing), while the `Pw_Array` parameter values represent the corresponding pulse widths at those points. You could think of the function as being analogous to a look-up table, where the input signal (`cntl` pin of the device) amplitude is mapped to the corresponding input value in the `cntl_array` and then the `Pw_Array` value that this is paired with, is used as the width for the output pulse signal.

The amplitude of the pulse is determined by the values assigned to the `Out_Low` and `Out_High` parameters.

The output pulse is controlled by means of the `clk` input. When this input reaches the level assigned to the `Clk_Trig` parameter, the pulse is triggered, on either the rising or falling edge of the clock, in accordance with the setting of the `Pos_Edge_Trig` parameter.

Upon triggering, the output reaches its high value after time:

```
Rise_Delay + Rise_Time
```

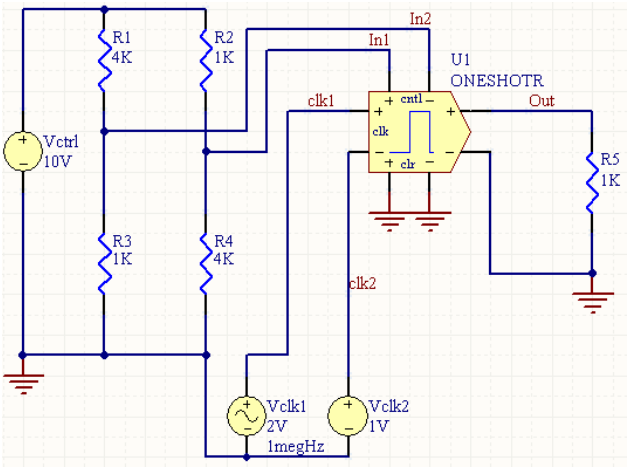
and its initial value again after time:

```
Pulse Width + Fall_Delay + Fall_Time
```

The `clr` input to the device is used to reset the state of the function so that it is possible to retrigger and thus obtain another pulse. The `clr` signal must be higher than the `Clk_Trig` signal to achieve this.

The input signal can be either a differential current or differential voltage signal.

Examples



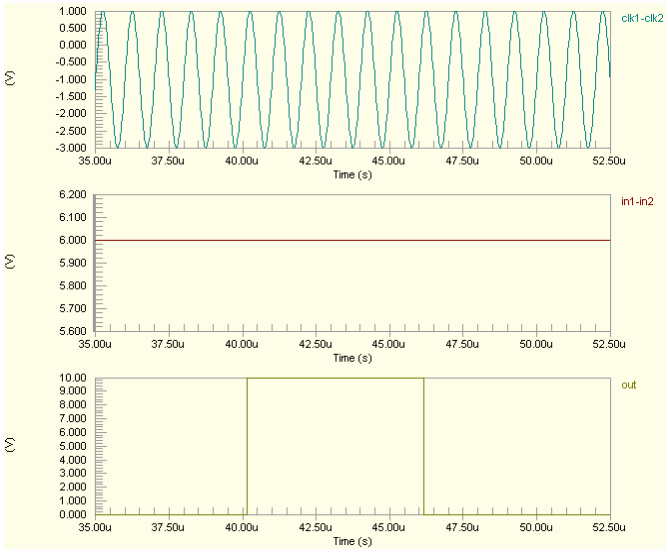
Consider the One-Shot function in the above image, with the following characteristics:

- Pin1 (positive clk input) is connected to net `clk1`
- Pin2 (negative clk input) is connected to net `clk2`
- Pin3 (positive cntl input) is connected to net `In1`
- Pin4 (negative cntl input) is connected to net `In2`
- Pin5 (positive clr input) is connected to net `GND`
- Pin6 (negative clr input) is connected to net `GND`
- Pin7 (positive output) is connected to net `Out`
- Pin8 (negative output) is connected to net `GND`
- Designator is `U1`
- cntl_array = `1 2 3 4 5 6 7 8 9 10 11`
- Pw_Array = `1u 2u 3u 4u 5u 6u 7u 8u 9u 10u 11u`
- Clk_Trig = `0.5`
- Out_High = `10`
- Out_Low = `0`
- Pos_Edge_Trig = `TRUE`
- Rise_Delay = `40u`.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU1 %vd(CLK1,CLK2) %vd(IN1,IN2) %vd(0,0) %vd(OUT,0) AU1ONESHOT
.MODEL AU1ONESHOT oneshot (cntl_array=[1 2 3 4 5 6 7 8 9 10 11] pw_array=[1u 2u
+ 3u 4u 5u 6u 7u 8u 9u 10u 11u] clk_trig=0.5 pos_edge_trig=TRUE out_low=0
+ out_high=10  rise_delay=40u   )
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

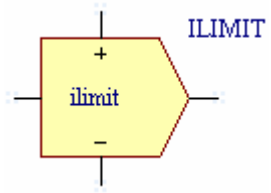


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `100.0u`
- **Transient Step Time** – set to `20.00n`
- **Transient Max Step Time** – set to `20.00n`.

Current Limiter

Current Limiter (Single-Ended I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

ILIMIT

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @"DESIGNATOR"ILIMIT
.MODEL @"DESIGNATOR"ILIMIT ilimit (?in_offset|in_offset=@in_offset| ?gain|gain=@gain|
?r_out_source|r_out_source=@r_out_source| ?r_out_sink|r_out_sink=@r_out_sink|
i_limit_source=@i_limit_source i_limit_sink=@i_limit_sink
?v_pwr_range|v_pwr_range=@v_pwr_range| ?i_source_range|i_source_range=@i_source_range|
?i_sink_range|i_sink_range=@i_sink_range| ?r_out_domain|r_out_domain=@r_out_domain|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

In_Offset	-	input offset (Default = 0).
Gain	-	gain (Default = 1).
R_Out_Source	-	sourcing resistance. The value entered must lie in the range 1.0e-9 to 1.0e9 (Default = 1).
R_Out_Sink	-	sinking resistance. The value entered must lie in the range 1.0e-9 to 1.0e9 (Default = 1).
I_Limit_Source	-	current sourcing limit. The value entered must be no lower than 1.0e-12.
I_Limit_Sink	-	current sinking limit. The value entered must be no lower than 1.0e-12.
V_Pwr_Range	-	upper and lower power supply smoothing range. The value entered must be no lower than 1.0e-15 (Default = 1.0e-6).
I_Source_Range	-	sourcing current smoothing range. The value entered must be no lower than 1.0e-15 (Default = 1.0e-9).
I_Sink_Range	-	sinking current smoothing range. The value entered must be no lower than 1.0e-15 (Default = 1.0e-9).
R_Out_Domain	-	internal/external voltage delta smoothing range. The value entered must be no lower than 1.0e-15 (Default = 1.0e-9).

Notes

This function models the operation of an operational amplifier or comparator, at the highest level. All of the device pins act as inputs, with three of the four - `pos-pwr`, `neg-pwr` and `out` - also capable of acting as outputs.

The device takes a single-ended voltage input and applies offset and gain, as determined by the values assigned to the `In_Offset` and `Gain` parameters. An equivalent internal voltage (V_{Eq}) is derived from the result, which is subsequently limited by the range defined by the voltages applied to the `pos_pwr` and `neg_pwr` pins. If V_{Eq} is greater than the subsequent voltage that appears on the `out` pin of the device, a sourcing current flows from the `out` pin. If the value for V_{Eq} is less than that seen on the `out` pin, a sinking current flows into the `out` pin.

If a sourcing current results, the value of the current will be controlled by a sourcing resistance, as defined by the `R_Out_Source` parameter. The sourcing current is limited to a maximum value defined by the `I_Limit_Source` parameter. The output current in this case will be reflected in the current through the `pos_pwr` pin of the device.

If a sinking current results, the value of the current will be controlled by a sinking resistance, as defined by the `R_Out_Sink` parameter. The sinking current is limited to a maximum value defined by the `I_Limit_Sink` parameter. (This models the limiting behavior found in the output stages of most operational amplifiers). The output current in this case will be reflected in the current through the `neg_pwr` pin of the device.

The `V_Pwr_Range` parameter is used to define the voltage level below V_{pos_pwr} and above V_{neg_pwr}, beyond which smoothing will be applied to the derived internal voltage signal, V_{Eq}.

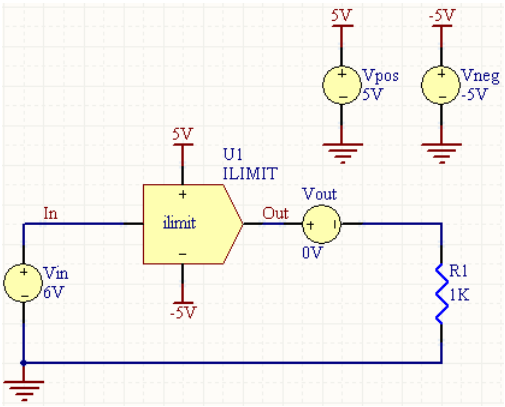
The `I_Source_Range` parameter is used to define the current level below `I_Limit_Source`, beyond which smoothing is applied. This value also determines the current increment above I_{out} = 0, at which the current through the `pos_pwr` pin begins to transition to zero.

The `I_Sink_Range` parameter is used to define the current level below `I_Limit_Sink`, beyond which smoothing is applied. This value also determines the current increment below I_{out} = 0, at which the current through the `neg_pwr` pin begins to transition to zero.

The `R_Out_Domain` parameter is used to specify the incremental value above and below (V_{Eq} - V_{out}) = 0, at which R_{out} will be set to `R_Out_Source` or `R_Out_Sink` respectively. R_{out} will be interpolated smoothly between `R_Out_Source` and `R_Out_Sink` under the following condition:

-R_Out_Domain < (V_{Eq} - V_{out}) > R_Out_Domain

Examples



Consider the current limiter in the above image, with the following characteristics:

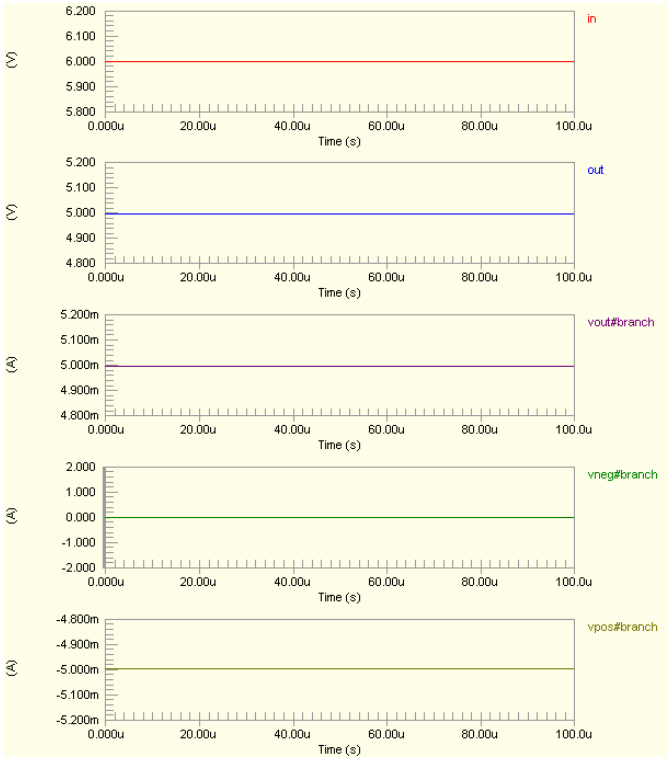
- Pin1 (in) is connected to net `In`
- Pin2 (pos_pwr) is connected to net `5V`
- Pin3 (neg_pwr) is connected to net `-5V`
- Pin4 (out) is connected to net `Out`
- Designator is `U1`
- Gain = `2`
- I_Limit_Source = `10mA`
- I_Limit_Sink = `10mA`
- All other model parameters are left at their inherent default values.

The entry in the SPICE netlist would be:

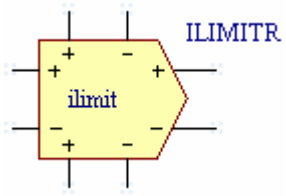
```
*Schematic Netlist:
AU1 IN 5V -5V OUT AU1ILIMIT
.MODEL AU1ILIMIT ilimit ( gain=2   i_limit_source=10M i_limit_sink=10m    )
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit. In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `100.0u`
- **Transient Step Time** – set to `20.00n`
- **Transient Max Step Time** – set to `20.00n`.



Current Limiter (Differential I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

ILIMIT

SPICE Netlist Template Format

```
@DESIGNATOR %%vd(%1,%2) %%gd(%3,%4) %%gd(%5,%6) %%gd(%7,%8) @"DESIGNATOR"ILIMIT
.MODEL @"DESIGNATOR"ILIMIT ilimit (?in_offset|in_offset=@in_offset| ?gain|gain=@gain|
?r_out_source|r_out_source=@r_out_source| ?r_out_sink|r_out_sink=@r_out_sink|
i_limit_source=@i_limit_source i_limit_sink=@i_limit_sink
?v_pwr_range|v_pwr_range=@v_pwr_range| ?i_source_range|i_source_range=@i_source_range|
?i_sink_range|i_sink_range=@i_sink_range| ?r_out_domain|r_out_domain=@r_out_domain|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

In_Offset	-	input offset (Default = 0).
Gain	-	gain (Default = 1).
R_Out_Source	-	sourcing resistance. The value entered must lie in the range 1.0e-9 to 1.0e9 (Default = 1).
R_Out_Sink	-	sinking resistance. The value entered must lie in the range 1.0e-9 to 1.0e9 (Default = 1).
I_Limit_Source	-	current sourcing limit. The value entered must be no lower than 1.0e-12.
I_Limit_Sink	-	current sinking limit. The value entered must be no lower than 1.0e-12.
V_Pwr_Range	-	upper and lower power supply smoothing range. The value entered must be no lower than 1.0e-15 (Default = 1.0e-6).
I_Source_Range	-	sourcing current smoothing range. The value entered must be no lower than 1.0e-15 (Default = 1.0e-9).
I_Sink_Range	-	sinking current smoothing range. The value entered must be no lower than 1.0e-15 (Default = 1.0e-9).
R_Out_Domain	-	internal/external voltage delta smoothing range. The value entered must be no lower than 1.0e-15 (Default = 1.0e-9).

Notes

This function models the operation of an operational amplifier or comparator, at the highest level. All of the device pins act as inputs, with six of the eight - `pos-pwr`, `neg-pwr` and `out` pin pairs (differential) - also capable of acting as outputs.

The device takes a differential voltage input and applies offset and gain, as determined by the values assigned to the `In_Offset` and `Gain` parameters. An equivalent internal voltage (V_{Eq}) is derived from the result, which is subsequently limited by the range defined by the differential voltages applied to the `pos_pwr` and `neg_pwr` pins. If V_{Eq} is greater than the subsequent voltage that appears on the `out` pin of the device, a sourcing current flows from the `out` pin. If the value for V_{Eq} is less than that seen on the `out` pin, a sinking current flows into the `out` pin.

If a sourcing current results, the value of the current will be controlled by a sourcing resistance, as defined by the `R_Out_Source` parameter. The sourcing current is limited to a maximum value defined by the `I_Limit_Source` parameter. The output current in this case will be reflected in the current through the `pos_pwr` pins of the device.

If a sinking current results, the value of the current will be controlled by a sinking resistance, as defined by the `R_Out_Sink` parameter. The sinking current is limited to a maximum value defined by the `I_Limit_Sink` parameter. (This models the limiting behavior found in the output stages of most operational amplifiers). The output current in this case will be reflected in the current through the `neg_pwr` pins of the device.

The `V_Pwr_Range` parameter is used to define the voltage level below V_{pos_pwr} and above V_{neg_pwr}, beyond which smoothing will be applied to the derived internal voltage signal, V_{Eq}.

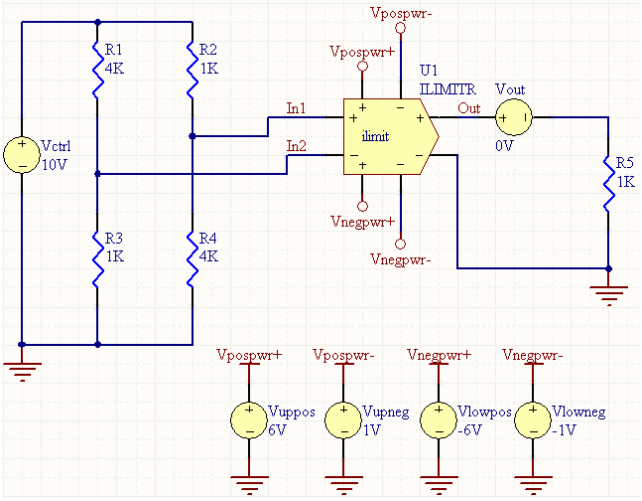
The `I_Source_Range` parameter is used to define the current level below `I_Limit_Source`, beyond which smoothing is applied. This value also determines the current increment above I_{out} = 0, at which the current through the `pos_pwr` pins begins to transition to zero.

The `I_Sink_Range` parameter is used to define the current level below `I_Limit_Sink`, beyond which smoothing is applied. This value also determines the current increment below I_{out} = 0, at which the current through the `neg_pwr` pins begins to transition to zero.

The `R_Out_Domain` parameter is used to specify the incremental value above and below (V_{Eq} - V_{out}) = 0, at which R_{out} will be set to `R_Out_Source` or `R_Out_Sink` respectively. R_{out} will be interpolated smoothly between `R_Out_Source` and `R_Out_Sink` under the following condition:

-R_Out_Domain < (V_{Eq} - V_{out}) > R_Out_Domain

Examples



Consider the current limiter in the above image, with the following characteristics:

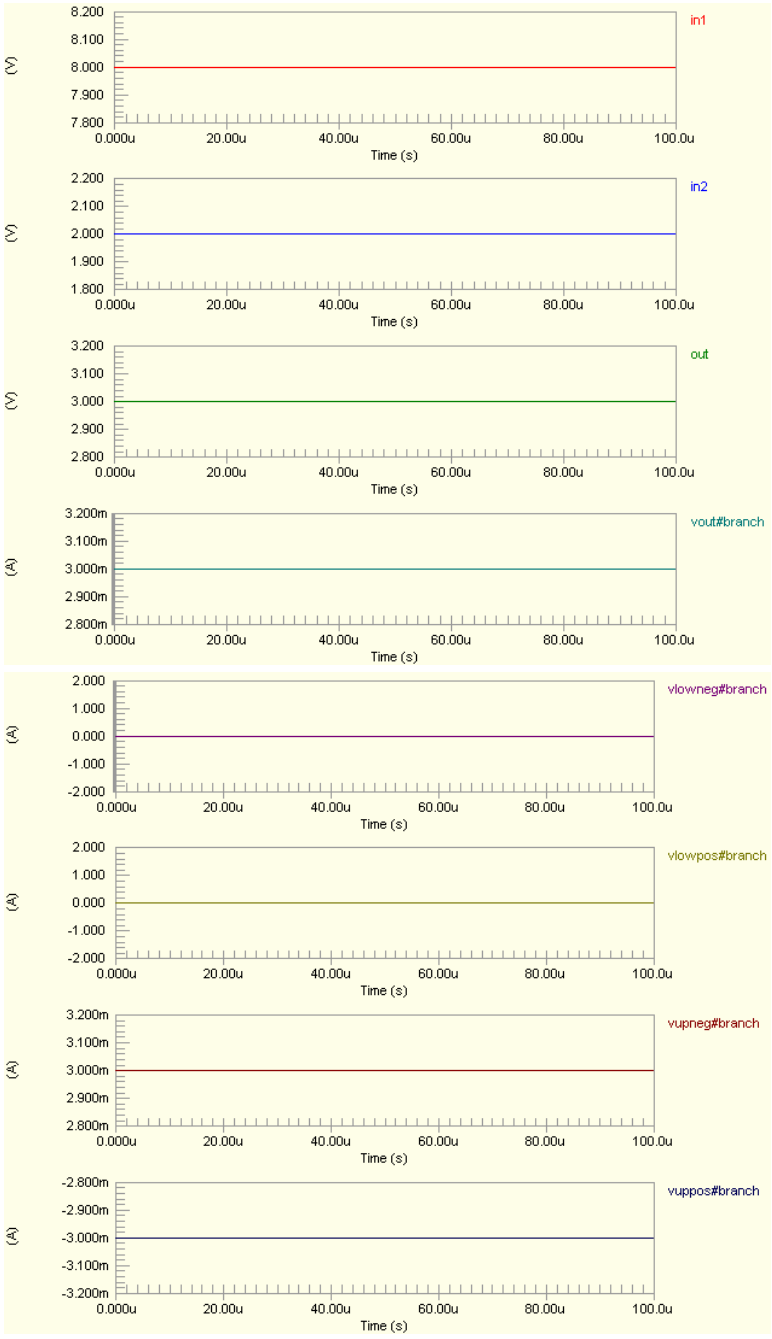
- Pin1 (positive input) is connected to net `In1`
- Pin2 (negative input) is connected to net `In2`
- Pin3 (positive pos_pwr input) is connected to net `Vpospwr+`
- Pin4 (negative pos_pwr input) is connected to net `Vpospwr-`
- Pin5 (positive neg_pwr input) is connected to net `Vnegpwr+`
- Pin6 (negative neg_pwr input) is connected to net `Vnegpwr-`
- pin7 (positive output) is connected to net `Out`
- Pin8 (negative output) is connected to net `GND`
- Designator is `U1`
- Gain = `2`
- I_Limit_Source = `3mA`
- I_Limit_Sink = `3mA`
- All other model parameters are left at their inherent default values.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU1 %vd(IN1,IN2) %gd(VPOSPWR+,VPOSPWR-) %gd(VNEGPWR+,VNEGPWR-) %gd(OUT,0)
+ AU1ILIMIT
.MODEL AU1ILIMIT ilimit ( gain=2   i_limit_source=3mA i_limit_sink=3mA    )
```

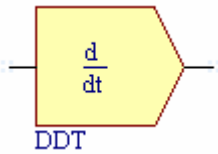
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit. In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `100.0u`
- **Transient Step Time** – set to `20.00n`
- **Transient Max Step Time** – set to `20.00n`.



Differentiator

Differentiator (Single-Ended I/0)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

D_DT

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @"DESIGNATOR"DDT
.MODEL @"DESIGNATOR"DDT d_dt (?out_offset|out_offset=@out_offset| ?gain|gain=@gain|
out_lower_limit=@out_lower_limit out_upper_limit=@out_upper_limit
?limit_range|limit_range=@limit_range|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Out_Offset	-	output offset (Default = 0).
Gain	-	gain (default = 1).
Out_Lower_Limit	-	output lower limit.
Out_Upper_Limit	-	output upper limit.
Limit_Range	-	upper and lower limit smoothing range. (Default = 1.0e-6).

Notes

This model is a simple derivative stage that approximates the time derivative of an input signal by calculating the incremental slope of that signal since the previous time point.

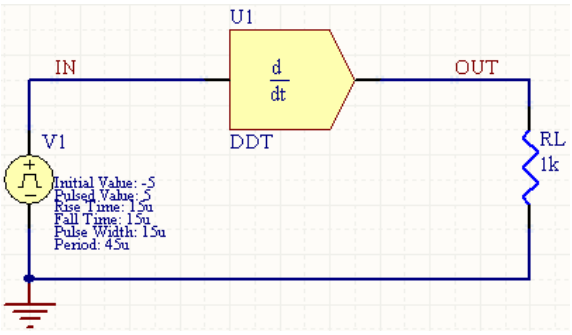
The output upper and lower limits are used to prevent convergence errors due to excessively high output values.

The `Limit_Range` specifies the value below `Out_Upper_Limit` and above `Out_Lower_Limit` at which smoothing of the output begins.

The Differentiator function does not include truncation error-checking. It is therefore not recommended that this function be used to provide integration through the use of a feedback loop. Undesirable results may be obtained. It is better in this case, to use the Integrator function, which provides for truncation error-checking.

The input signal can be either a single-ended current or single-ended voltage signal.

Examples



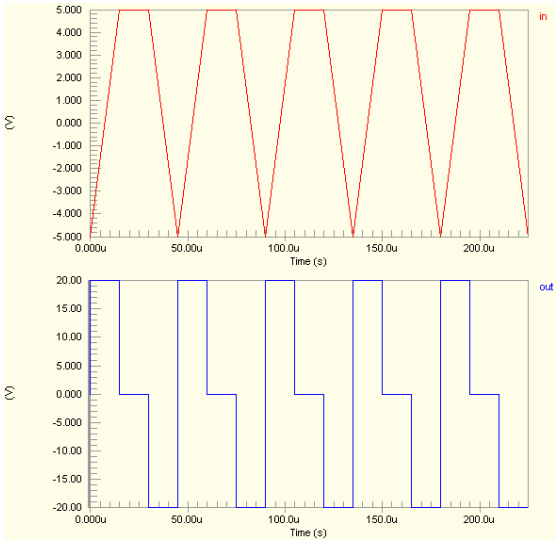
Consider the differentiator in the above image, with the following characteristics:

- Pin1 (in) is connected to net `IN`
- Pin2 (out) is connected to net `OUT`
- Designator is `U1`
- Out_Lower_Limit = `-20`
- Out_Upper_Limit = `20`
- All other parameters are left at their default values.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU1 IN OUT AU1DDT
.MODEL AU1DDT d_dt (  out_lower_limit=-20 out_upper_limit=20 )
```

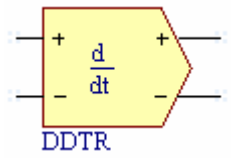
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `225.0u`
- **Transient Step Time** – set to `900.0n`
- **Transient Max Step Time** – set to `900.0n`.

Differentiator (Differential I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

D_DT

SPICE Netlist Template Format

```
@DESIGNATOR %%vd(%1,%2) %%vd(%3,%4) @"DESIGNATOR"DDT
.MODEL @"DESIGNATOR"DDT d_dt (?out_offset|out_offset=@out_offset| ?gain|gain=@gain|
out_lower_limit=@out_lower_limit out_upper_limit=@out_upper_limit
?limit_range|limit_range=@limit_range|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Out_Offset	-	output offset (Default = 0).
Gain	-	gain (default = 1).
Out_Lower_Limit	-	output lower limit.
Out_Upper_Limit	-	output upper limit.
Limit_Range	-	upper and lower limit smoothing range. (Default = 1.0e-6).

Notes

This model is a simple derivative stage that approximates the time derivative of an input signal by calculating the incremental slope of that signal since the previous time point.

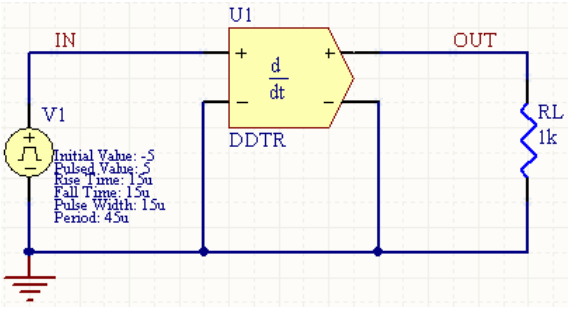
The output upper and lower limits are used to prevent convergence errors due to excessively high output values.

The `Limit_Range` specifies the value below `Out_Upper_Limit` and above `Out_Lower_Limit` at which smoothing of the output begins.

The Differentiator function does not include truncation error-checking. It is therefore not recommended that this function be used to provide integration through the use of a feedback loop. Undesirable results may be obtained. It is better in this case, to use the Integrator function, which provides for truncation error-checking.

The input signal can be either a differential current or differential voltage signal.

Examples



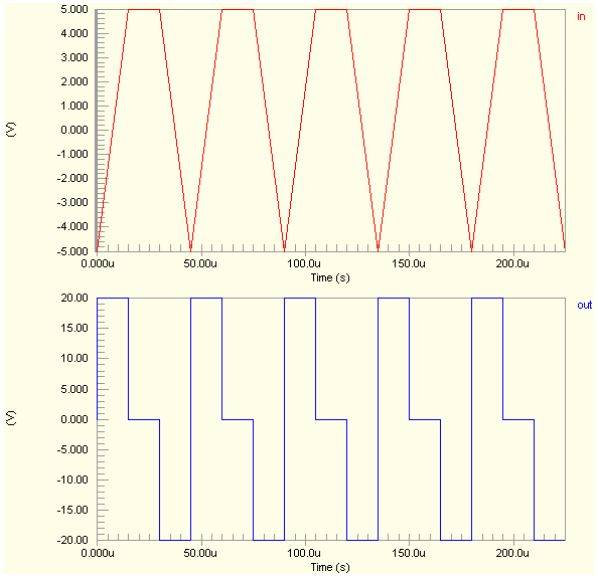
Consider the differentiator function in the above image, with the following characteristics:

- Pin1 (positive input) is connected to net `IN`
- Pin2 (negative input) is connected to net `GND`
- Pin3 (positive output) is connected to net `OUT`
- Pin4 (negative output) is connected to net `GND`
- Designator is `U1`
- Out_Lower_Limit =`-20`
- Out_Upper_Limit = `20`
- All other parameters are left at their default values.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU1 %vd(IN,0) %vd(OUT,0) AU1DDT
.MODEL AU1DDT d_dt (  out_lower_limit=-20 out_upper_limit=20 )
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

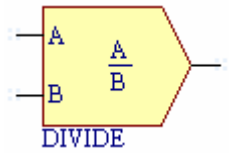


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `225.0u`
- **Transient Step Time** – set to `900.0n`
- **Transient Max Step Time** – set to `900.0n`.

Divider

Divider (Single-Ended I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

DIVIDE

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 @"DESIGNATOR"DIVIDE
.MODEL @"DESIGNATOR"DIVIDE divide (?num_offset|num_offset=@num_offset|
?num_gain|num_gain=@num_gain| ?den_offset|den_offset=@den_offset|
?den_gain|den_gain=@den_gain| ?den_lower_limit|den_lower_limit=@den_lower_limit|
?den_domain|den_domain=@den_domain| ?fraction|fraction=@fraction|
?out_gain|out_gain=@out_gain| ?out_offset|out_offset=@out_offset|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Num_Offset	-	numerator offset (Default = 0).
Num_Gain	-	numerator gain (Default = 1).
Den_Offset	-	denominator offset (Default = 0).
Den_Gain	-	denominator gain (Default = 1).
Den_Lower_Limit	-	denominator lower limit (Default = 1.0e-10).
Den_Domain	-	denominator smoothing domain (Default = 1.0e-10).
Fraction	-	used to control whether the smoothing domain is specified as a fractional (TRUE) or absolute (FALSE) value. (Default = FALSE).
Out_Gain	-	output gain (Default = 1).
Out_Offset	-	output offset (Default = 0).

Notes

This is a two-quadrant divider. It takes two inputs (one specified as the numerator, the other as the denominator) and processes them to obtain the output result as follows:

- The inputs are offset, in accordance with the values specified for the `Num_Offset` and `Den_Offset` parameters.
- The offset signals are then multiplied by the values for gain specified in the respective `Num_Gain` and `Den_Gain` parameters.
- The resulting values are divided
- The quotient is multiplied by the value specified for the `Out_Gain` parameter.
- The output result is then offset in accordance with the value specified for the `Out_Offset` parameter.

The process can be expressed mathematically as follows:

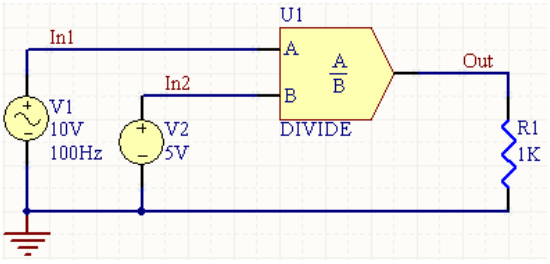
Output = ((((Num + Num_Offset) * Num_Gain) / ((Den + Den_Offset) * Den_Gain)) * Out_Gain) + Out_Offset

The denominator is prevented from ever going zero by specification of a limiting positive value in the `Den_Lower_Limit` parameter. This limit is reached through the use of a quadratic smoothing function, the domain of which is specified using the `Den_Domain` parameter.

This model will operate in DC, AC and Transient analysis modes only. When running an AC Small Signal analysis, the results are only valid when one of the two inputs, not both, is connected to an AC signal.

The input signals can be either single-ended current or single-ended voltage signals.

Examples



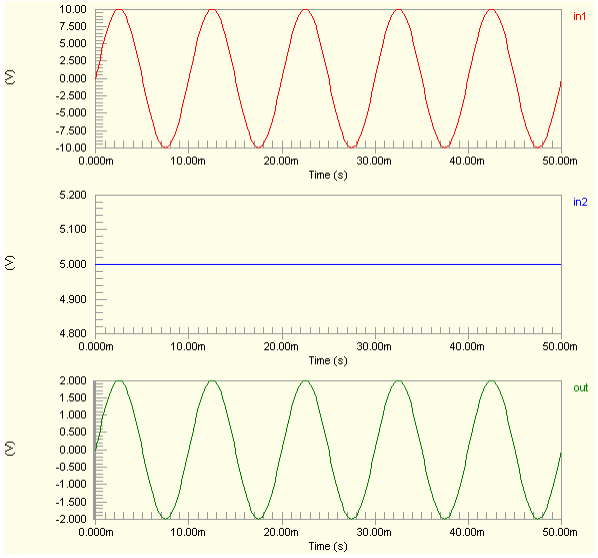
Consider the divider in the above image, with the following characteristics:

- Pin1 (num) is connected to net `In1`
- Pin2 (den) is connected to net `In2`
- Pin3 (out) is connected to net `Out`
- Designator is `U1`
- All parameters are left at their default values

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU1 IN1 IN2 OUT AU1DIVIDE
.MODEL AU1DIVIDE divide ()
```

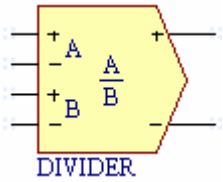
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Divider (Differential I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

DIVIDE

SPICE Netlist Template Format

```
@DESIGNATOR %%vd(%1,%2) %%vd(%3,%4) %%vd(%5,%6) @"DESIGNATOR"DIVIDE
.MODEL @"DESIGNATOR"DIVIDE divide (?num_offset|num_offset=@num_offset|
?num_gain|num_gain=@num_gain| ?den_offset|den_offset=@den_offset|
?den_gain|den_gain=@den_gain| ?den_lower_limit|den_lower_limit=@den_lower_limit|
?den_domain|den_domain=@den_domain| ?fraction|fraction=@fraction|
?out_gain|out_gain=@out_gain| ?out_offset|out_offset=@out_offset|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Num_Offset - numerator offset (Default = 0).

Num_Gain - numerator gain (Default = 1).

Den_Offset - denominator offset (Default = 0).

Den_Gain - denominator gain (Default = 1).

Den_Lower_Limit - denominator lower limit (Default = 1.0e-10).

Den_Domain - denominator smoothing domain (Default = 1.0e-10).

Fraction - used to control whether the smoothing domain is specified as a fractional (TRUE) or absolute (FALSE) value. (Default = FALSE).

Out_Gain - output gain (Default = 1).

Out_Offset - output offset (Default = 0).

Notes

This is a two-quadrant divider. It takes two inputs (one specified as the numerator, the other as the denominator) and processes them to obtain the output result as follows:

- The inputs are offset, in accordance with the values specified for the `Num_Offset` and `Den_Offset` parameters.
- The offset signals are then multiplied by the values for gain specified in the respective `Num_Gain` and `Den_Gain` parameters.
- The resulting values are divided
- The quotient is multiplied by the value specified for the `Out_Gain` parameter.
- The output result is then offset in accordance with the value specified for the `Out_Offset` parameter.

The process can be expressed mathematically as follows:

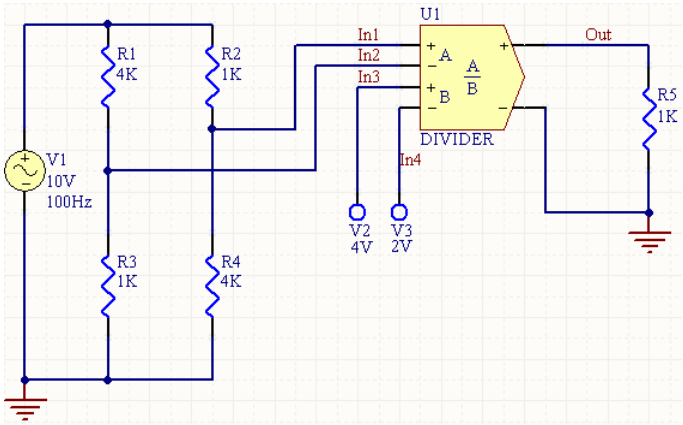
Output = ((((Num + Num_Offset) * Num_Gain) / ((Den + Den_Offset) * Den_Gain)) * Out_Gain) + Out_Offset

The denominator is prevented from ever going zero by specification of a limiting positive value in the `Den_Lower_Limit` parameter. This limit is reached through the use of a quadratic smoothing function, the domain of which is specified using the `Den_Domain` parameter.

This model will operate in DC, AC and Transient analysis modes only. When running an AC Small Signal analysis, the results are only valid when one of the two inputs, not both, is connected to an AC signal.

The input signals can be either differential current or differential voltage signals.

Examples



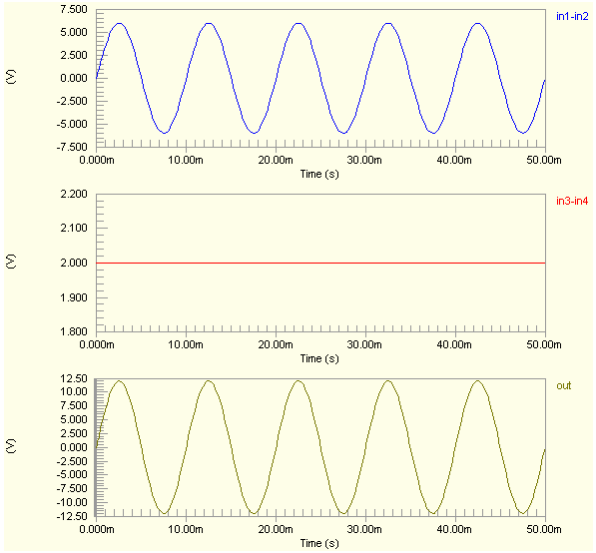
Consider the divider in the above image, with the following characteristics:

- Pin1 (positive num input) is connected to net `In1`
- Pin2 (negative num input) is connected to net `In2`
- Pin3 (positive den input) is connected to net `In3`
- Pin4 (negative den input) is connected to net `In4`
- Pin5 (positive output) is connected to net `Out`
- Pin6 (negative output) is connected to net `GND`
- Designator is `U1`
- Out_Gain = `4`
- All other model parameters are left at their inherent default values

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU1 %vd(IN1,IN2) %vd(IN3,IN4) %vd(OUT,0) AU1DIVIDE
.MODEL AU1DIVIDE divide (out_gain=4)
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

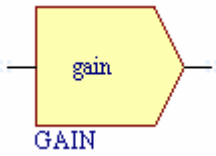


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Gain

Gain (Single-Ended I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

GAIN

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @"DESIGNATOR"GAIN
.MODEL @"DESIGNATOR"GAIN gain (?in_offset|in_offset=@in_offset| ?gain|gain=@gain|
?out_offset|out_offset=@out_offset|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

In_Offset - input offset (Default = 0).

Gain - gain (Default = 1).

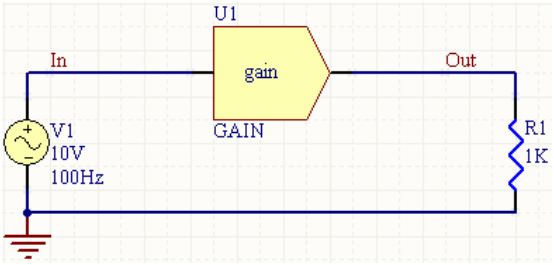
Out_Offset - output offset (Default = 0).

Notes

This is a simple gain block that takes the input signal and multiplies it by the value assigned to the `Gain` parameter. Optional offset adjusts are available on both input and output.

The input signal can be either a single-ended current or single-ended voltage signal.

Examples



Consider the gain function in the above image, with the following characteristics:

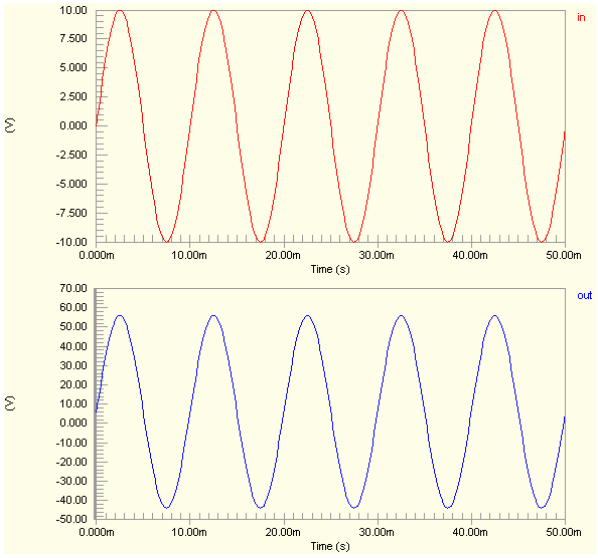
- Pin1 (input) is connected to net `In`
- Pin2 (output) is connected to net `Out`
- Designator is `U1`
- In_Offset = `2V`
- Gain = `5`

- Out_Offset = `-4V`

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU1 IN OUT AU1GAIN
.MODEL AU1GAIN gain (in_offset=2V gain=5 out_offset=-4V)
```

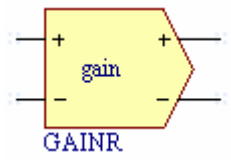
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Gain (Differential I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

GAIN

SPICE Netlist Template Format

```
@DESIGNATOR %%vd(%1,%2) %%vd(%3,%4) @"DESIGNATOR"GAIN
.MODEL @"DESIGNATOR"GAIN gain (?in_offset|in_offset=@in_offset| ?gain|gain=@gain|
?out_offset|out_offset=@out_offset|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

In_Offset - input offset (Default = 0).

Gain - gain (Default = 1).

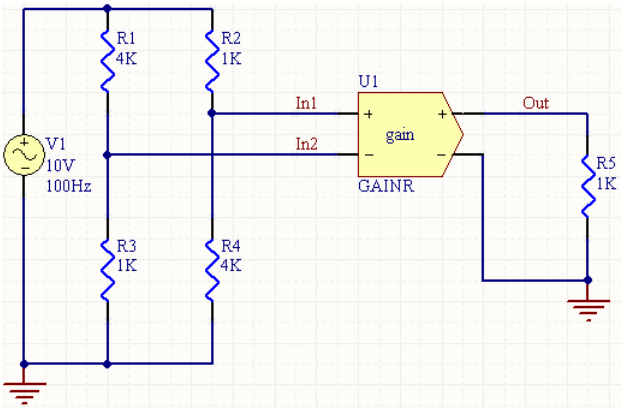
Out_Offset - output offset (Default = 0).

Notes

This is a simple gain block that takes the input signal and multiplies it by the value assigned to the `Gain` parameter. Optional offset adjusts are available on both input and output.

The input signal can be either a differential current or differential voltage signal.

Examples



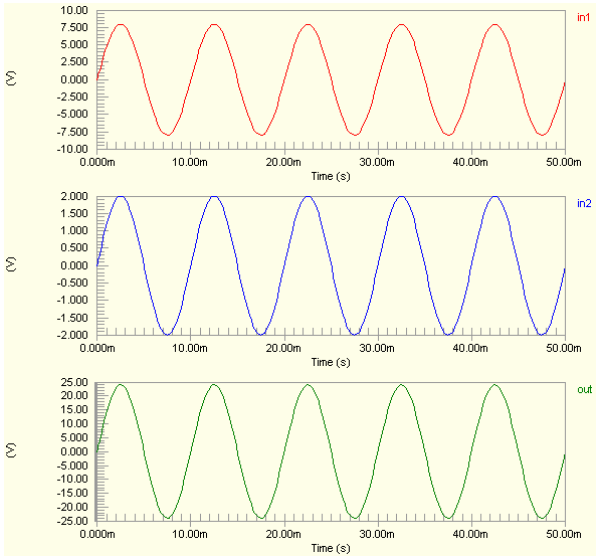
Consider the gain function in the above image, with the following characteristics:

- Pin1 (positive input) is connected to net `In1`
- Pin2 (negative input) is connected to net `In2`
- Pin3 (positive output) is connected to net `Out`
- Pin4 (negative output) is connected to net `GND`
- Designator is `U1`
- Gain =`4`
- All other model parameters are left at their inherent default values.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU1 %vd(IN1,IN2) %vd(OUT,0) AU1GAIN
.MODEL AU1GAIN gain ( gain=4 )
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

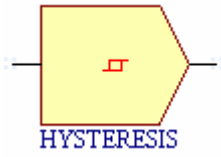


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Hysteresis

Hysteresis (Single-Ended I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

HYST

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @"DESIGNATOR"HYST
.MODEL @"DESIGNATOR"HYST hyst (?in_low|in_low=@in_low| ?in_high|in_high=@in_high|
?hyst|hyst=@hyst| ?out_lower_limit|out_lower_limit=@out_lower_limit|
?out_upper_limit|out_upper_limit=@out_upper_limit| ?input_domain|input_domain=@input_domain|
?fraction|fraction=@fraction|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

In_Low	-	input low value (Default = 0).
In_High	-	input high value (Default = 1).
Hyst	-	hysteresis. The value entered must be a positive real number (Default = 0.1).
Out_Lower_Limit	-	output lower limit (Default = 0).
Out_Upper_Limit	-	output upper limit (Default = 1).
Input_Domain	-	input smoothing domain (Default = 0.01).
Fraction	-	used to control whether the smoothing domain is specified as a fractional (TRUE) or absolute (FALSE) value. (Default = TRUE).

Notes

This is a simple buffer stage, providing hysteresis of the output with respect to the input. The input points about which the hysteresis effect operates is determined by the values assigned to the `In_Low` and `In_High` parameters.

The output is limited by the specification of the `Out_Lower_Limit` and `Out_Upper_Limit` parameters.

The points at which the hysteresis slope would normally change abruptly are defined as:

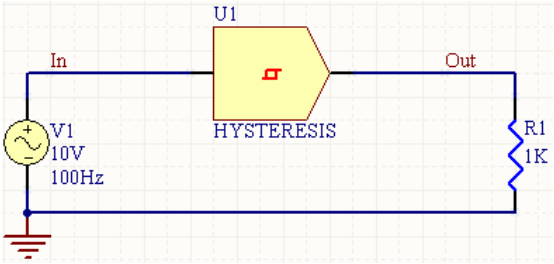
In_Low + Hyst and **In_High + Hyst** for input transition from low to high.

In_Low - Hyst and **In_High - Hyst** for input transition from high to low.

Use of the `Input_Domain` parameter, with a positive value, ensures that the hysteresis slope never changes abruptly, but is rather smoothed over the specified domain (the region prior to the hysteresis slope meeting the defined limit level).

The input signal can be either a single-ended current or single-ended voltage signal.

Examples



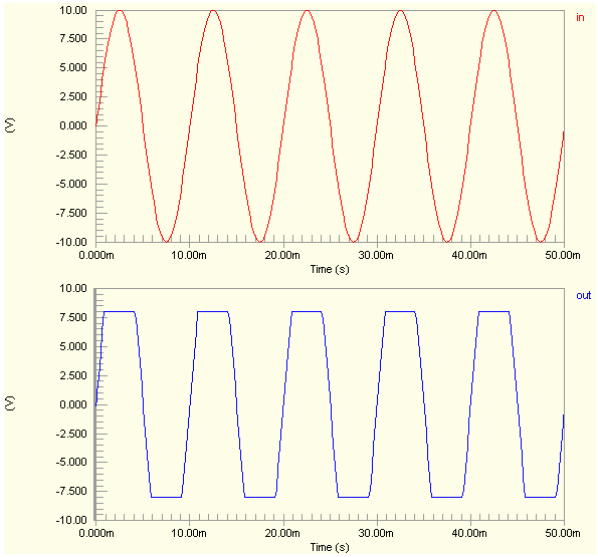
Consider the hysteresis function in the above image, with the following characteristics:

- Pin1 (input) is connected to net `In`
- Pin2 (output) is connected to net `Out`
- Designator is `U1`
- In_Low = `-5V`
- In_High = `5V`
- Out_Lower_Limit = `-8V`
- Out_Upper_Limit = `8V`
- All other model parameters are left at their inherent defaults.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU1 IN OUT AU1HYST
.MODEL AU1HYST hyst (in_low=-5 in_high=5  out_lower_limit=-8 out_upper_limit=8  )
```

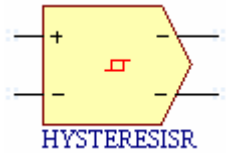
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Hysteresis (Differential I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

HYST

SPICE Netlist Template Format

```
@DESIGNATOR %%vd(%1,%2) %%vd(%3,%4) @"DESIGNATOR"HYST
.MODEL @"DESIGNATOR"HYST hyst (?in_low|in_low=@in_low| ?in_high|in_high=@in_high|
?hyst|hyst=@hyst| ?out_lower_limit|out_lower_limit=@out_lower_limit|
?out_upper_limit|out_upper_limit=@out_upper_limit| ?input_domain|input_domain=@input_domain|
?fraction|fraction=@fraction|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

In_Low - input low value (Default = 0).

In_High	-	input high value (Default = 1).
Hyst	-	hysteresis. The value entered must be a positive real number (Default = 0.1).
Out_Lower_Limit	-	output lower limit (Default = 0).
Out_Upper_Limit	-	output upper limit (Default = 1).
Input_Domain	-	input smoothing domain (Default = 0.01).
Fraction	-	used to control whether the smoothing domain is specified as a fractional (TRUE) or absolute (FALSE) value. (Default = TRUE).

Notes

This is a simple buffer stage, providing hysteresis of the output with respect to the input. The input points about which the hysteresis effect operates is determined by the values assigned to the `In_Low` and `In_High` parameters.

The output is limited by the specification of the `Out_Lower_Limit` and `Out_Upper_Limit` parameters.

The points at which the hysteresis slope would normally change abruptly are defined as:

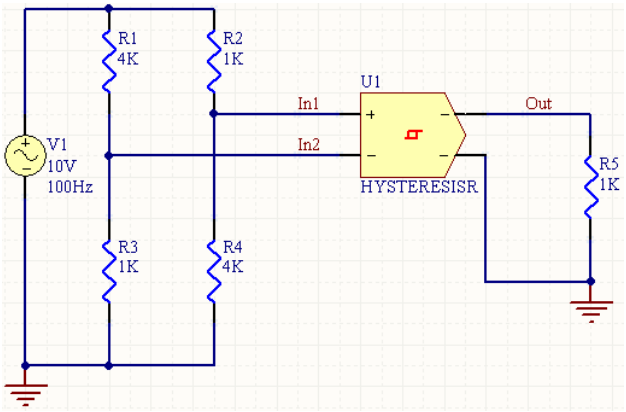
In_Low + Hyst and **In_High + Hyst** for input transition from low to high.

In_Low - Hyst and **In_High - Hyst** for input transition from high to low.

Use of the `Input_Domain` parameter, with a positive value, ensures that the hysteresis slope never changes abruptly, but is rather smoothed over the specified domain (the region prior to the hysteresis slope meeting the defined limit level).

The input signal can be either a differential current or differential voltage signal.

Examples



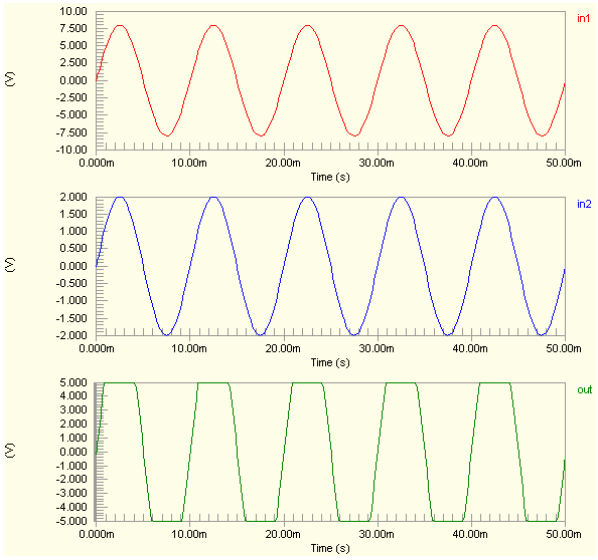
Consider the hysteresis function in the above image, with the following characteristics:

- Pin1 (positive input) is connected to net `In1`
- Pin2 (negative input) is connected to net `In2`
- Pin3 (positive output) is connected to net `Out`
- Pin4 (negative output) is connected to net `GND`
- Designator is `U1`
- In_Low = `-3V`
- In_High = `3V`
- Out_Lower_Limit = `-5V`
- Out_Upper_Limit = `5V`
- All other model parameters are left at their inherent defaults.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU1 %vd(IN1,IN2) %vd(OUT,0) AU1HYST
.MODEL AU1HYST hyst (in_low=-3 in_high=3  out_lower_limit=-5 out_upper_limit=5  )
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

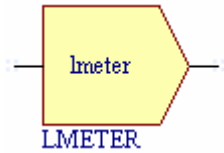


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Inductance Meter

Inductance Meter (Single-Ended I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

LMETER

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @"DESIGNATOR"LMETER
.MODEL @"DESIGNATOR"LMETER lmeter (?gain|gain=@gain|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

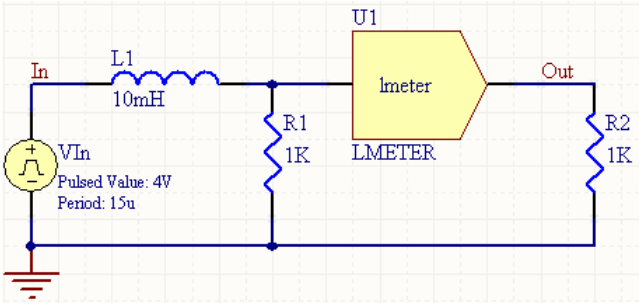
Gain - gain (default = 1).

Notes

This is a sensing device which is attached to a node in the circuit and produces as an output, a scaled value equal to the total inductance seen on its input, multiplied by the value assigned to the `Gain` parameter. This model is useful as a building block for other models, which require to sense an inductance value and adjust their behavior with respect to it.

The input signal can be either a single-ended current or single-ended voltage signal.

Examples



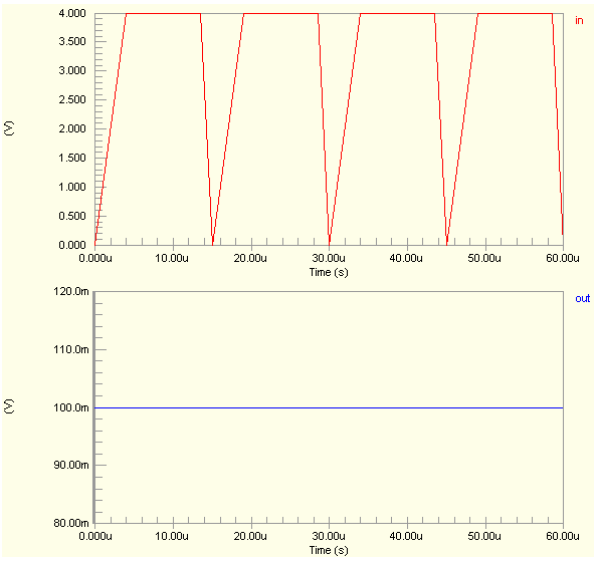
Consider the inductance meter in the above image, with the following characteristics:

- Pin1 (input) is connected to net `NetL1_2`
- Pin2 (output) is connected to net `Out`
- Designator is `U1`
- Gain = `10`

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU1 NetL1_2 OUT AU1LMETER
.MODEL AU1LMETER lmeter (gain=10)
```

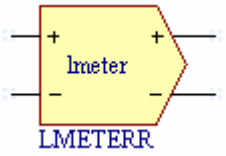
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `60.00u`
- **Transient Step Time** – set to `2.000u`
- **Transient Max Step Time** – set to `2.000u`.

Inductance Meter (Differential I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

LMETER

SPICE Netlist Template Format

```
@DESIGNATOR %%vd(%1,%2) %%vd(%3,%4) @"DESIGNATOR"LMETER
.MODEL @"DESIGNATOR"LMETER lmeter (?gain|gain=@gain|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

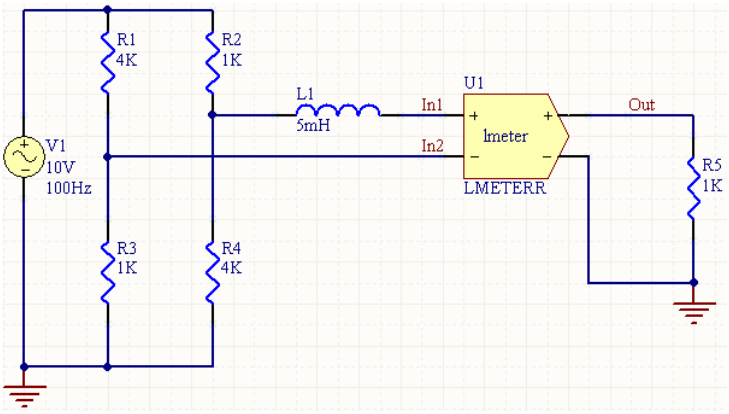
Gain - gain (default = 1).

Notes

This is a sensing device which is attached to a node in the circuit and produces as an output, a scaled value equal to the total inductance seen on its input, multiplied by the value assigned to the `Gain` parameter. This model is useful as a building block for other models, which require to sense an inductance value and adjust their behavior with respect to it.

The input signal can be either a differential current or differential voltage signal.

Examples



Consider the inductance meter in the above image, with the following characteristics:

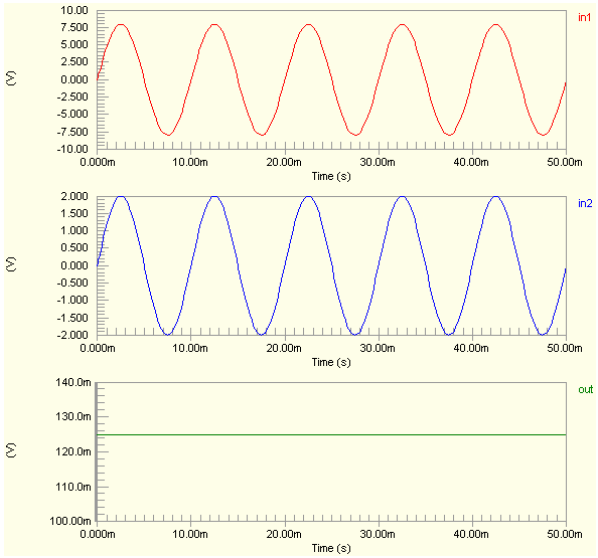
- Pin1 (positive input) is connected to net `In1`
- Pin2 (negative input) is connected to net `In2`
- Pin3 (positive output) is connected to net `Out`
- Pin4 (negative output) is connected to net `GND`
- Designator is `U1`

- Gain = `25`

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU1 %vd(IN1,IN2) %vd(OUT,0) AU1LMETER
.MODEL AU1LMETER lmeter (gain=25)
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

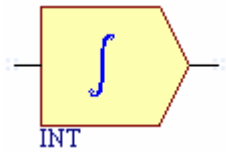


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Integrator

Integrator (Single-Ended I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

INT

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @"DESIGNATOR"INT
.MODEL @"DESIGNATOR"INT int (?in_offset|in_offset=@in_offset| ?gain|gain=@gain|
out_lower_limit=@out_lower_limit out_upper_limit=@out_upper_limit
?limit_range|limit_range=@limit_range| ?out_ic|out_ic=@out_ic|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

In_Offset	-	input offset (Default = 0).
Gain	-	gain (Default = 1).
Out_Lower_Limit	-	output lower limit.
Out_Upper_Limit	-	output upper limit.
Limit_Range	-	upper and lower limit smoothing range (Default = 1.0e-6).
Out_IC	-	output initial condition (Default = 0).

Notes

This model is a simple integration stage that approximates the integral of the input with respect to time.

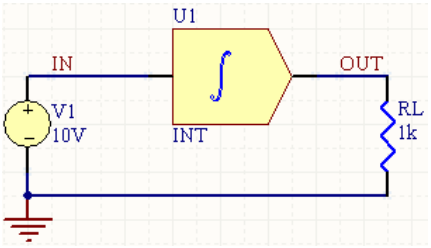
The output upper and lower limits are used to prevent convergence errors due to excessively high output values. These limits provide for integrator behavior similar to that found in the integration stage of an operational amplifier. Once a limit has been reached, no further storage of values occurs.

The `Limit_Range` specifies the value below `Out_Upper_Limit` and above `Out_Lower_Limit` at which smoothing of the output begins.

Truncation error checking is an inherent part of the model. If truncation errors become excessive, the model uses smaller time increments between simulation data points, therefore providing for a more accurate simulation of the integration function.

The input signal can be either a single-ended current or single-ended voltage signal.

Examples



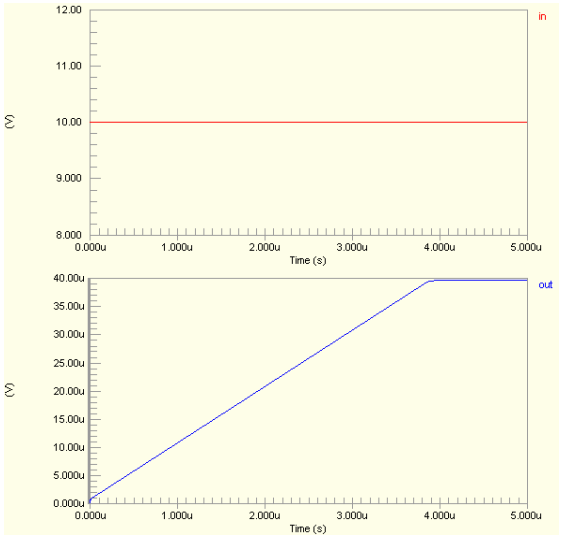
Consider the integrator in the above image, with the following characteristics:

- Pin1 (in) is connected to net `IN`
- Pin2 (out) is connected to net `OUT`
- Designator is `U1`
- Out_Lower_Limit = `0`
- Out_Upper_Limit = `40e-6`
- All other parameters are left at their default values.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU1 IN OUT AU1INT
.MODEL AU1INT int (  out_lower_limit=0 out_upper_limit=40e-6 )
```

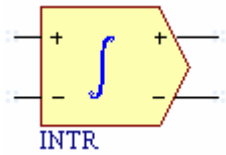
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `5.000u`
- **Transient Step Time** – set to `20.00n`
- **Transient Max Step Time** – set to `20.00n`.

Integrator (Differential I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

INT

SPICE Netlist Template Format

```
@DESIGNATOR %%vd(%1,%2) %%vd(%3,%4) @"DESIGNATOR"INT
.MODEL @"DESIGNATOR"INT int (?in_offset|in_offset=@in_offset| ?gain|gain=@gain|
out_lower_limit=@out_lower_limit out_upper_limit=@out_upper_limit
?limit_range|limit_range=@limit_range| ?out_ic|out_ic=@out_ic|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

In_Offset - input offset (Default = 0).

Gain	-	gain (Default = 1).
Out_Lower_Limit	-	output lower limit.
Out_Upper_Limit	-	output upper limit.
Limit_Range	-	upper and lower limit smoothing range (Default = 1.0e-6).
Out_IC	-	output initial condition (Default = 0).

Notes

This model is a simple integration stage that approximates the integral of the input with respect to time.

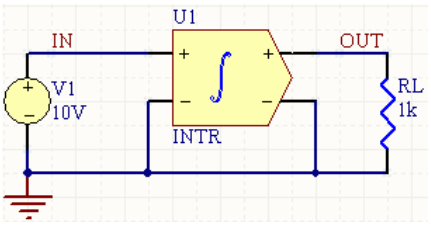
The output upper and lower limits are used to prevent convergence errors due to excessively high output values. These limits provide for integrator behavior similar to that found in the integration stage of an operational amplifier. Once a limit has been reached, no further storage of values occurs.

The `Limit_Range` specifies the value below `Out_Upper_Limit` and above `Out_Lower_Limit` at which smoothing of the output begins.

Truncation error checking is an inherent part of the model. If truncation errors become excessive, the model uses smaller time increments between simulation data points, therefore providing for a more accurate simulation of the integration function.

The input signal can be either a differential current or differential voltage signal.

Examples



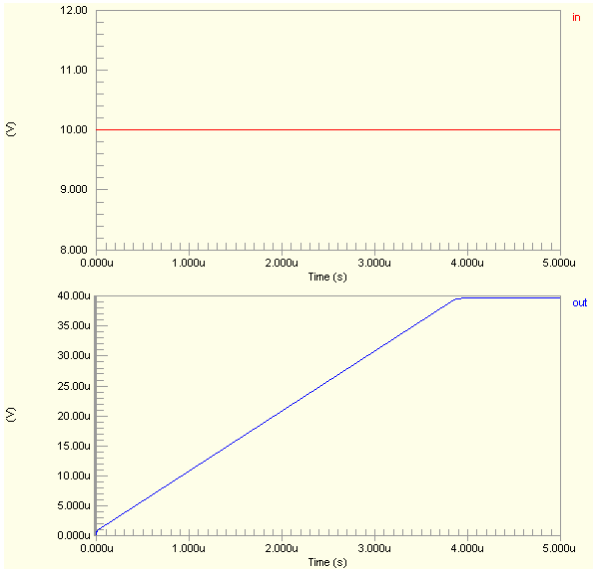
Consider the integrator function in the above image, with the following characteristics:

- Pin1 (positive input) is connected to net `IN`
- Pin2 (negative input) is connected to net `GND`
- Pin3 (positive output) is connected to net `OUT`
- Pin4 (negative output) is connected to net `GND`
- Designator is `U1`
- Out_Lower_Limit = `0`
- Out_Upper_Limit = `40e-6`
- All other parameters are left at their default values.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU1 %vd(IN,0) %vd(OUT,0) AU1INT
.MODEL AU1INT int (  out_lower_limit=0 out_upper_limit=40e-6 )
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

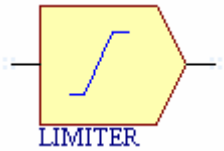


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `5.000u`
- **Transient Step Time** – set to `20.00n`
- **Transient Max Step Time** – set to `20.00n`.

Limiter

Limiter (Single-Ended I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

LIMIT

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @"DESIGNATOR"LIMIT
.MODEL @"DESIGNATOR"LIMIT limit (?in_offset|in_offset=@in_offset| ?gain|gain=@gain|
?out_lower_limit|out_lower_limit=@out_lower_limit|
?out_upper_limit|out_upper_limit=@out_upper_limit| ?limit_range|limit_range=@limit_range|
?fraction|fraction=@fraction|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

In_Offset	-	input offset (Default = 0).
Gain	-	gain (Default = 1).
Out_Lower_Limit	-	output lower limit (Default = 0).
Out_Upper_Limit	-	output upper limit (Default = 1).
Limit_Range	-	upper and lower smoothing range (Default = 1.0e-6).
Fraction	-	used to control whether the is specified as a fractional (TRUE) or absolute (FALSE) value. (Default = FALSE).

Notes

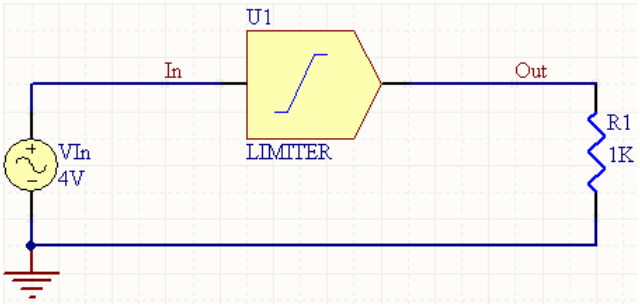
This model is similar in function to the Gain function. However, the output is restricted to the range specified by the output lower and upper limits. The input signal can be either a single-ended current or single-ended voltage signal.

This model is also similar in function to the Controlled Limiter, the difference being that the output limiting is defined using parameters of the model, rather than providing the limit levels external to the device.

The `Limit_Range` is the value below `Out_Upper_Limit` and above `Out_Lower_Limit` at which smoothing of the output begins. The `Limit_Range` therefore represents the delta, with respect to the output level, at which smoothing occurs. For example, for an input `Gain` of 2, `Limit_Range` of 0.1V and output limits of 1V (upper) and -1V (lower), the output will begin to smooth out at +/-0.9 V.

When the `Limit_Range` is specified as a fractional value (`Fraction` parameter set to TRUE), it is expressed as the calculated fraction of the difference between `Out_Upper_Limit` and `Out_Lower_Limit`.

Examples



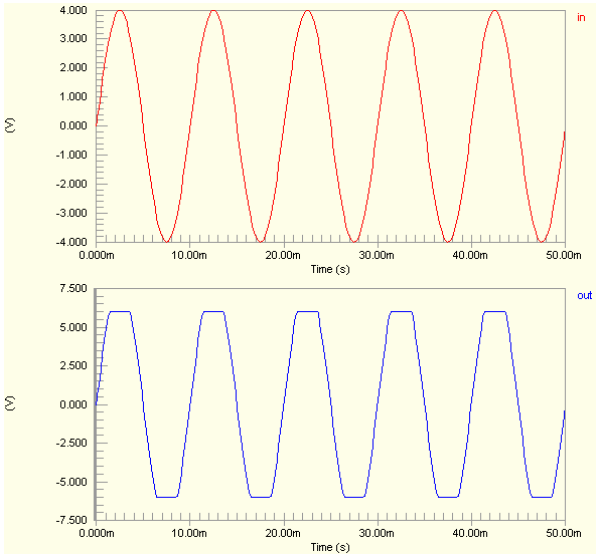
Consider the limiter in the above image, with the following characteristics:

- Pin1 (input) is connected to net `In`
- Pin2 (output) is connected to net `Out`
- Designator is `U1`
- Gain = `2`
- Out_Lower_Limit = `-6V`
- Out_Upper_Limit = `6V`
- Limit_Range = `0.1V`
- All other model parameters are left at their inherent defaults.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU1 IN OUT AU1LIMIT
.MODEL AU1LIMIT limit ( gain=2 out_lower_limit=-6 out_upper_limit=6
+ limit_range=0.1 )
```

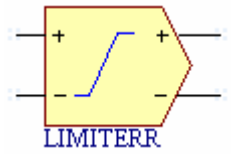
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Limiter (Differential I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

LIMIT

SPICE Netlist Template Format

```
@DESIGNATOR %%vd(%1,%2) %%vd(%3,%4) @"DESIGNATOR"LIMIT
.MODEL @"DESIGNATOR"LIMIT limit (?in_offset|in_offset=@in_offset| ?gain|gain=@gain|
?out_lower_limit|out_lower_limit=@out_lower_limit|
?out_upper_limit|out_upper_limit=@out_upper_limit| ?limit_range|limit_range=@limit_range|
?fraction|fraction=@fraction|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

In_Offset - input offset (Default = 0).

Gain	-	gain (Default = 1).
Out_Lower_Limit	-	output lower limit (Default = 0).
Out_Upper_Limit	-	output upper limit (Default = 1).
Limit_Range	-	upper and lower smoothing range (Default = 1.0e-6).
Fraction	-	used to control whether the is specified as a fractional (TRUE) or absolute (FALSE) value. (Default = FALSE).

Notes

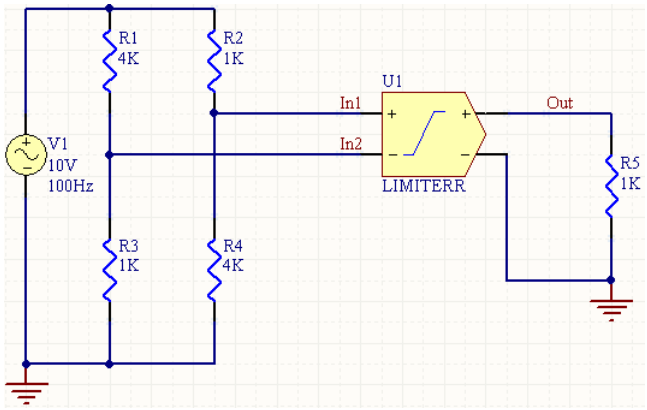
This model is similar in function to the Gain function. However, the output is restricted to the range specified by the output lower and upper limits. The input signal can be either a differential current or differential voltage signal.

This model is also similar in function to the Controlled Limiter, the difference being that the output limiting is defined using parameters of the model, rather than providing the limit levels external to the device.

The `Limit_Range` is the value below `Out_Upper_Limit` and above `Out_Lower_Limit` at which smoothing of the output begins. The `Limit_Range` therefore represents the delta, with respect to the output level, at which smoothing occurs. For example, for an input `Gain` of 2, `Limit_Range` of 0.1V and output limits of 1V (upper) and -1V (lower), the output will begin to smooth out at +/-0.9 V.

When the `Limit_Range` is specified as a fractional value (`Fraction` parameter set to TRUE), it is expressed as the calculated fraction of the difference between `Out_Upper_Limit` and `Out_Lower_Limit`.

Examples



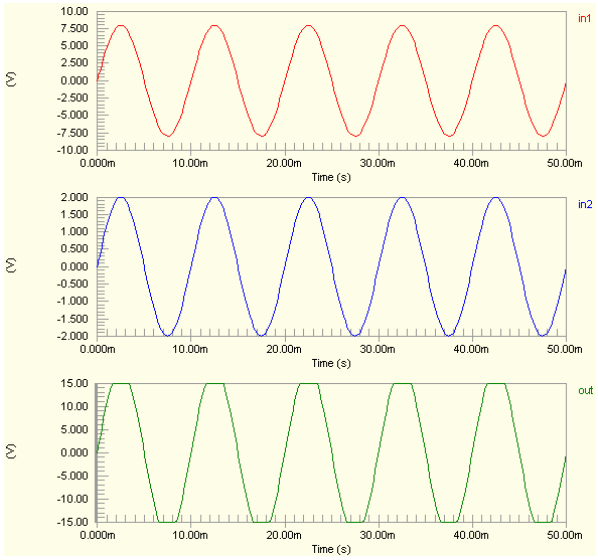
Consider the limiter in the above image, with the following characteristics:

- Pin1 (positive input) is connected to net `In1`
- Pin2 (negative input) is connected to net `In2`
- Pin3 (positive output) is connected to net `Out`
- Pin4 (negative output) is connected to net `GND`
- Designator is `U1`
- Gain = `3`
- Out_Lower_Limit = `-15V`
- Out_Upper_Limit = `15V`
- All other model parameters are left at their inherent defaults.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU1 %vd(IN1,IN2) %vd(OUT,0) AU1LIMIT
.MODEL AU1LIMIT limit ( gain=3 out_lower_limit=-15 out_upper_limit=15  )
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

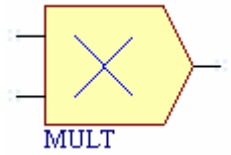


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Multiplier

Multiplier (Single-Ended I/O)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

MULT

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 @MODEL #|PARAMS:| ?X_OFFSET|X_OFFSET=@X_OFFSET|
?Y_OFFSET|Y_OFFSET=@Y_OFFSET| ?X_GAIN|X_GAIN=@X_GAIN| ?Y_GAIN|Y_GAIN=@Y_GAIN|
?OUT_GAIN|OUT_GAIN=@OUT_GAIN| ?OUT_OFFSET|OUT_OFFSET=@OUT_OFFSET|
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

X_Offset - X input offset (Default = 0).

Y_Offset - Y input offset (Default = 0).

X_Gain - X input gain (Default = 1).

Y_Gain - Y input gain (Default = 1).

Out_Gain - output gain (Default = 1).

Out_Offset - output offset (Default = 0).

Notes

This is a two-input multiplier with offset and gain adjustment available on both inputs and output. It takes the inputs and processes them to obtain the output result as follows:

- The inputs are offset, in accordance with the values specified for the `X_Offset` and `Y_Offset` parameters.
- The offset signals are then multiplied by the values for gain specified in the respective `X_Gain` and `Y_Gain` parameters.
- The resulting values are multiplied
- The result is then multiplied by the value specified for the `Out_Gain` parameter.
- The output result is then offset in accordance with the value specified for the `Out_Offset` parameter.

The process can be expressed mathematically as follows:

Output = (((X + X_Offset) * X_Gain) * ((Y + Y_Offset) * Y_Gain) * Out_Gain) + Out_Offset

This model will operate in DC, AC and Transient analysis modes only. When running an AC Small Signal analysis, the results are only valid when one of the two inputs, not both, is connected to an AC signal.

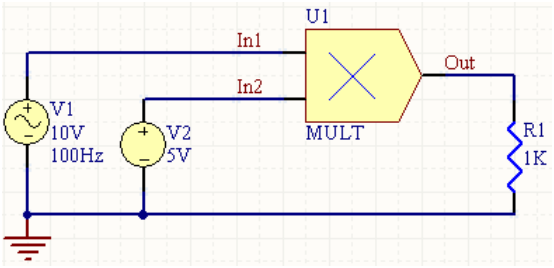
The input signals can be either single-ended current or single-ended voltage signals.

The built-in XSpice multiplier function can take two or more inputs, with no upper limit on the number of inputs considered. This particular 2-input version is defined using the hierarchical sub-circuit syntax. Within the sub-circuit definition, the XSpice Multiplier model is called and the parameters of the sub-circuit file parsed to this model.

Entering a value for a parameter on the **Parameters** tab of the *Sim Model* dialog will override its specified value in the sub-circuit file.

To check the default values of the supplied 2-input multiplier, open the appropriate sub-circuit (`.ckt`) file. You can view the content of this file for the model specified on the **Model Kind** tab of the *Sim Model* dialog, by clicking on the **Model File** tab at the bottom of the dialog. The default parameter values are listed in the `.SUBCKT` line.

Examples



Consider the multiplier in the above image, with the following characteristics:

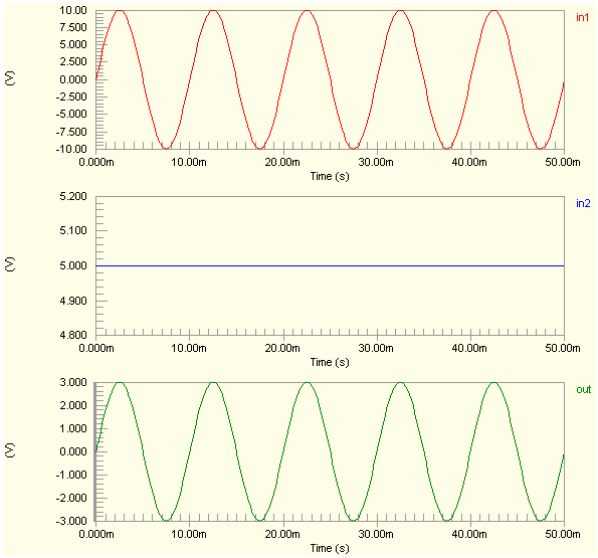
- Pin1 (in_a) is connected to net `In1`
- Pin2 (in_b) is connected to net `In2`
- Pin3 (out) is connected to net `Out`
- Designator is `U1`
- X_Gain = `2` (defined on **Parameters** tab)
- Y_Gain = `3` (defined on **Parameters** tab)
- Out_Gain = `0.01` (defined on **Parameters** tab)
- All other parameters are left at their default values.

The entries in the SPICE netlist would be:

```
*Schematic Netlist:
XU1 IN1 IN2 OUT MULT PARAMS:   x_gain=2 y_gain=3 out_gain=0.01
```

```
.
.
*Models and Subcircuit:
.SUBCKT MULT 1  2  3 PARAMS: x_offset=0.0 y_offset=0.0 x_gain=1.0 y_gain=1.0
+ out_gain=1.0 out_offset=0.0
A1 [1 2] 3 sigmult
.model sigmult mult(in_offset=[{x_offset} {y_offset}] in_gain=[{x_gain} {y_gain}]
+ out_gain={out_gain} out_offset={out_offset})
.ENDS MULT
```

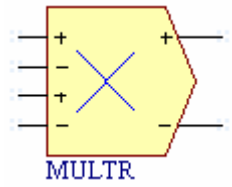
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Multiplier (Differential I/O)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

MULTR

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 %5 %6 @MODEL #|PARAMS:| ?x_offset|x_offset=@x_offset|
?y_offset|y_offset=@y_offset| ?x_gain|x_gain=@x_gain| ?y_gain|y_gain=@y_gain|
?out_gain|out_gain=@out_gain| ?out_offset|out_offset=@out_offset|
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

X_Offset - X input offset (Default = 0).

Y_Offset - Y input offset (Default = 0).

X_Gain - X input gain (Default = 1).

Y_Gain - Y input gain (Default = 1).

Out_Gain - output gain (Default = 1).

Out_Offset - output offset (Default = 0).

Notes

This is a two-input multiplier with offset and gain adjustment available on both inputs and output. It takes the inputs and processes them to obtain the output result as follows:

- The inputs are offset, in accordance with the values specified for the `X_Offset` and `Y_Offset` parameters.
- The offset signals are then multiplied by the values for gain specified in the respective `X_Gain` and `Y_Gain` parameters.
- The resulting values are multiplied
- The result is then multiplied by the value specified for the `Out_Gain` parameter.
- The output result is then offset in accordance with the value specified for the `Out_Offset` parameter.

The process can be expressed mathematically as follows:

Output = (((X + X_Offset) * X_Gain) * ((Y + Y_Offset) * Y_Gain) * Out_Gain) + Out_Offset

This model will operate in DC, AC and Transient analysis modes only. When running an AC Small Signal analysis, the results are only valid when one of the two inputs, not both, is connected to an AC signal.

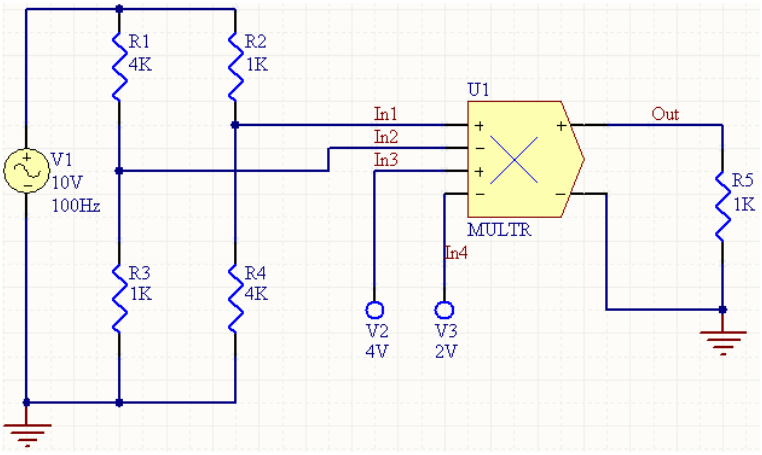
The input signals can be either differential current or differential voltage signals.

The built-in XSpice multiplier function can take two or more inputs, with no upper limit on the number of inputs considered. This particular 2-input version is defined using the hierarchical sub-circuit syntax. Within the sub-circuit definition, the XSpice Multiplier model is called and the parameters of the sub-circuit file parsed to this model.

Entering a value for a parameter on the **Parameters** tab of the *Sim Model* dialog will override its specified value in the sub-circuit file.

To check the default values of the supplied 2-input multiplier, open the appropriate sub-circuit (`.ckt`) file. You can view the content of this file for the model specified on the **Model Kind** tab of the *Sim Model* dialog, by clicking on the **Model File** tab at the bottom of the dialog. The default parameter values are listed in the `.SUBCKT` line.

Examples



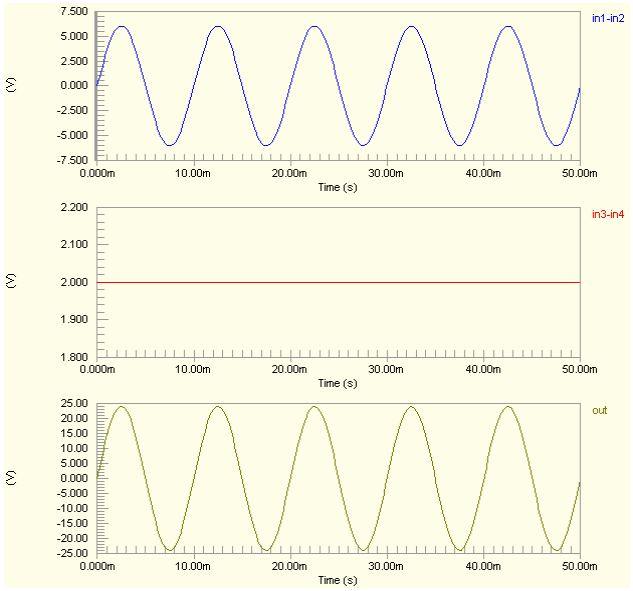
Consider the multiplier in the above image, with the following characteristics:

- Pin1 (positive a input) is connected to net `In1`
- Pin2 (negative a input) is connected to net `In2`
- Pin3 (positive b input) is connected to net `In3`
- Pin4 (negative b input) is connected to net `In4`
- Pin5 (positive output) is connected to net `Out`
- Pin6 (negative output) is connected to net `GND`
- Designator is `U1`
- X_Gain = `0.5` (defined on the **Parameters** tab)
- Y_Gain = `2` (defined on the **Parameters** tab)
- Out_Gain = `2` (defined on the **Parameters** tab)
- All other parameters are left at their default values.

The entries in the SPICE netlist would be:

```
*Schematic Netlist:
XU1 IN1 IN2 IN3 IN4 OUT 0 MULTR PARAMS:   x_gain=0.5 y_gain=2 out_gain=2
.
.
*Models and Subcircuit:
.SUBCKT MULTR 1   2   3   4   5   6 PARAMS: x_offset=0.0 y_offset=0.0 x_gain=1.0
+ y_gain=1.0 out_gain=1.0 out_offset=0.0
A1 [%vd(1,2) %vd(3,4)] %vd(5,6) sigmult
.model sigmult mult(in_offset=[{x_offset} {y_offset}] in_gain=[{x_gain} {y_gain}]
+ out_gain={out_gain} out_offset={out_offset})
.ENDS MULTR
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

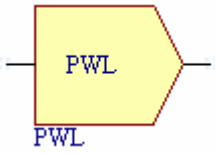


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

PWL Controlled Source

PWL Controlled Source (Single-Ended I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

PWL

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @"DESIGNATOR"PWL
.MODEL @"DESIGNATOR"PWL pwl (x_array =[@x_array] y_array =[@y_array]
?input_domain|input_domain=@input_domain| ?fraction|fraction=@fraction|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

x_array - x-element array. Enter a list of progressively increasing values, using spaces as separators. At

		least two values must be entered for the array.
y_array	-	y-element array. Enter a list of values, using spaces as separators. At least two values must be entered for the array.
input_domain	-	input smoothing domain. Enter a value in the range 1.0e-12 to 0.5 (Default = 0.01).
fraction	-	used to control whether the smoothing domain is specified as a fractional (TRUE) or absolute (FALSE) value. (Default = TRUE).

Notes

The function of this model is to take the input signal and provide an output that is dependent on a piece-wise linear waveform, as defined by coordinate values specified in the `x_array` and `y_array` parameters.

The `x_array` parameter values are input coordinate points (progressively increasing), while the `y_array` parameter values represent the corresponding outputs at those points. You could think of the function as being analogous to a look-up table, where the input signal amplitude is mapped to the corresponding input value in the `x_array` and then the `y_array` value that this is paired with, is used for the output signal.

For values of the input signal that are smaller than the first element value of `x_array` and greater than that of the last, the function uses the lowest and highest two coordinate pairs respectively and extends the slope between each. The function is therefore perfectly linear before the first coordinate and after the last coordinate specified by the arrays.

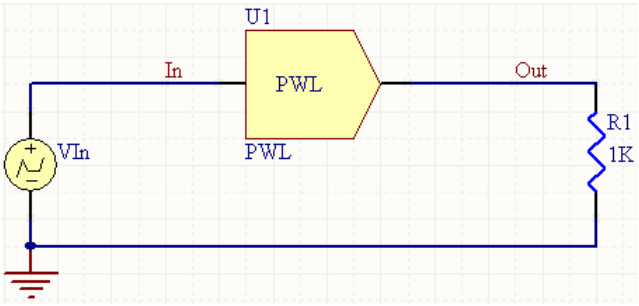
The PWL function does not have inherent output limiting. Care should therefore be taken as it is quite possible to end up with excessively large or small outputs for larger values of input.

The use of the smoothing domain around each coordinate point in the defined PWL waveform, reduces the possibility of non-convergence. Inherent checking of the value entered for the `input_domain` parameter is carried out by the model, so that overlap of smoothing domains does not result from too high a value being specified.

Care should be taken when using the smoothing domain as a fractional value (`fraction` = TRUE), as excessive smoothing can result if the coordinates specified in the `x_array` and `y_array` parameters are inappropriate.

The input signal can be either a single-ended current or single-ended voltage signal.

Examples



Consider the PWL function in the above image, with the following characteristics:

- Pin1 (input) is connected to net `In`
- Pin2 (output) is connected to net `Out`
- Designator is `U1`
- x_array = `0 1 2 3 4 5 6 7 8 9 10`
- y_array = `0 0 0 5 10 10 10 5 0 0 0`
- input_domain = `1e-3`
- fraction = `FALSE`.

The `Time/Value Pairs` for the Piecewise-Linear Voltage Source are:

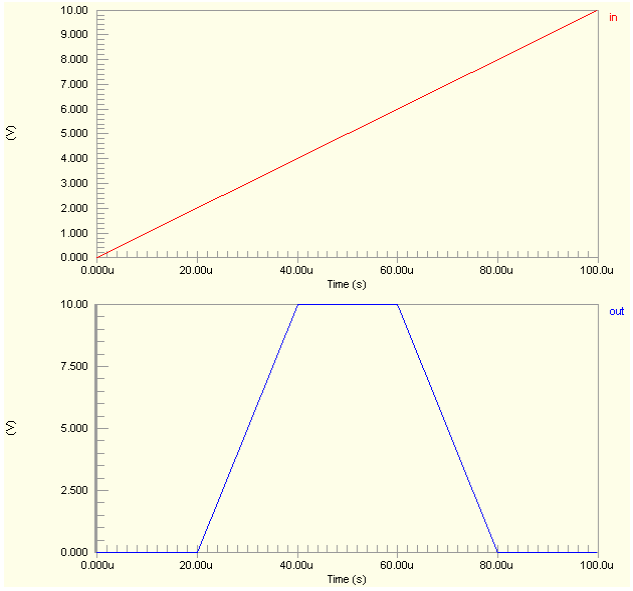
```
0U 0V
10U 1V
20U 2V
30U 3V
40U 4V
```

```
50U 5V
60U 6V
70U 7V
80U 8V
90U 9V
100U 10V.
```

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU1 IN OUT AU1PWL
.MODEL AU1PWL pwl (x_array =[0 1 2 3 4 5 6 7 8 9 10] y_array =[0 0 0 5 10 10 10 5
+ 0 0 0] input_domain=1e-3 fraction=FALSE)
```

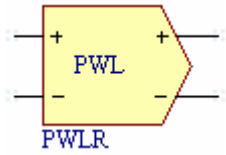
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `100.0u`
- **Transient Step Time** – set to `400.0n`
- **Transient Max Step Time** – set to `400.0n`.

PWL Controlled Source (Differential I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

PWL

SPICE Netlist Template Format

```
@DESIGNATOR %%vd(%1,%2) %%vd(%3,%4)@"DESIGNATOR"PWL
.MODEL @"DESIGNATOR"PWL pwl (x_array =[@x_array] y_array =[@y_array]
?input_domain|input_domain=@input_domain| ?fraction|fraction=@fraction|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

x_array - x-element array. Enter a list of progressively increasing values, using spaces as separators. At least two values must be entered for the array.

y_array - y-element array. Enter a list of values, using spaces as separators. At least two values must be entered for the array.

input_domain - input smoothing domain. Enter a value in the range 1.0e-12 to 0.5 (Default = 0.01).

fraction - used to control whether the smoothing domain is specified as a fractional (TRUE) or absolute (FALSE) value. (Default = TRUE).

Notes

The function of this model is to take the input signal and provide an output that is dependent on a piece-wise linear waveform, as defined by coordinate values specified in the `x_array` and `y_array` parameters.

The `x_array` parameter values are input coordinate points (progressively increasing), while the `y_array` parameter values represent the corresponding outputs at those points. You could think of the function as being analogous to a look-up table, where the input signal amplitude is mapped to the corresponding input value in the `x_array` and then the `y_array` value that this is paired with, is used for the output signal.

For values of the input signal that are smaller than the first element value of `x_array` and greater than that of the last, the function uses the lowest and highest two coordinate pairs respectively and extends the slope between each. The function is therefore perfectly linear before the first coordinate and after the last coordinate specified by the arrays.

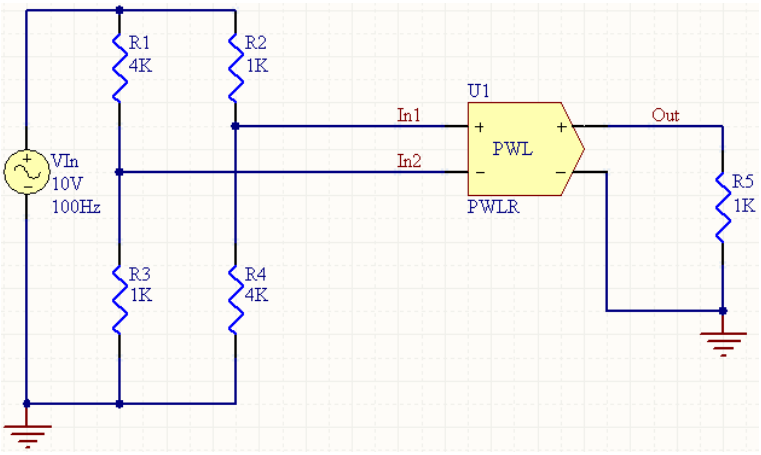
The PWL function does not have inherent output limiting. Care should therefore be taken as it is quite possible to end up with excessively large or small outputs for larger values of input.

The use of the smoothing domain around each coordinate point in the defined PWL waveform, reduces the possibility of non-convergence. Inherent checking of the value entered for the `input_domain` parameter is carried out by the model, so that overlap of smoothing domains does not result from too high a value being specified.

Care should be taken when using the smoothing domain as a fractional value (`fraction` = TRUE), as excessive smoothing can result if the coordinates specified in the `x_array` and `y_array` parameters are inappropriate.

The input signal can be either a differential current or differential voltage signal.

Examples



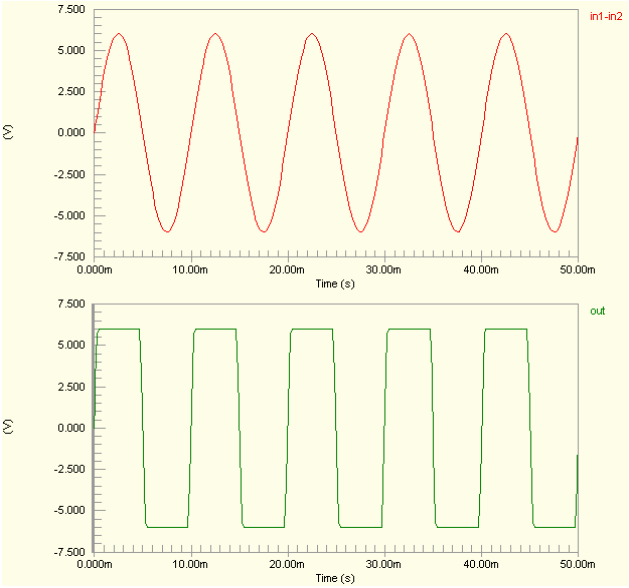
Consider the PWL function in the above image, with the following characteristics:

- Pin1 (positive input) is connected to net `In1`
- Pin2 (negative input) is connected to net `In2`
- Pin3 (positive output) is connected to net `Out`
- Pin4 (negative output) is connected to net `GND`
- Designator is `U1`
- x_array = `-6 -5 -4 -3 -2 -1 0 1 2 3 4 5 6`
- y_array = `-6 -6 -6 -6 -6 -6 0 6 6 6 6 6 6`
- input_domain = `0.1`
- fraction = `FALSE`.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU1 %vd(IN1,IN2) %vd(OUT,0)AU1PWL
.MODEL AU1PWL pwl (x_array =[-6 -5 -4 -3 -2 -1 0 1 2 3 4 5 6] y_array =[-6 -6 -6
+ -6 -6 -6 0 6 6 6 6 6 6] input_domain=0.1 fraction=FALSE)
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



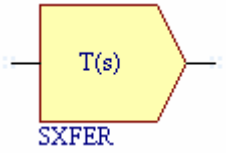
In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`

- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

S-Domain Transfer Function

S-Domain Transfer Function (Single-Ended I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

S_XFER

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @"DESIGNATOR"SXFER
.MODEL @"DESIGNATOR"SXFER s_xfer (?in_offset|in_offset=@in_offset| ?gain|gain=@gain|
num_coeff=[@num_coeff] den_coeff=[@den_coeff] ?int_ic|int_ic=[@int_ic]|
?denormalized_freq|denormalized_freq=@denormalized_freq|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

in_offset	-	input offset (Default = 0).
gain	-	gain (Default = 1).
num_coeff	-	numerator polynomial coefficients. Enter a list of values, using spaces as separators. At least one value must be entered for the array.
den_coeff	-	denominator polynomial coefficients. Enter a list of values, using spaces as separators. At least one value must be entered for the array.
int_ic	-	integrator stage initial conditions. (Default = 0).
denormalized_freq	-	denormalized corner frequency (in radians per second). This allows you to specify the coefficients for a normalized filter, where the frequency of interest is 1 rad/s, and then move the corner frequency to the one of interest (denormalizing the transfer function). (Default = 1).

Notes

This model provides a single input, single output transfer function in the Laplace transform variable, **s**. This function enables you to modulate the frequency-domain characteristics of a signal.

The s-domain transfer function you define must adhere to the following two restrictions:

- The degree of the numerator polynomial cannot exceed that of the denominator polynomial.
- All polynomial coefficients must be stated explicitly, even if a coefficient is zero.

The model takes the single-ended input signal, applies any offset and gain specified by the `in_offset` and `gain` parameters and then multiplies the result by the transfer function determined by the polynomial coefficient entered in the respective `num_coeff` and `den_coeff` parameters.

When specifying the coefficients for numerator and denominator, the highest powered term coefficient must be entered first, followed by those coefficients for subsequent decreasing power terms.

There are no limits on the internal signal values, or on the output of the transfer function. Care should therefore be taken when specifying coefficients and gain, so that excessively large output values do not result.

In AC Small Signal analysis, the output of the function is equal to the real and imaginary components of the total s-domain transfer function at each frequency of interest.

The `int_ic` parameter is an array that must be the same size as the array of values specified for the `den_coeff` parameter. For example, if there are three coefficient entries defined in the `den_coeff` parameter, then the `int_ic` parameter must also have three entries, using spaces as separators. By default, this parameter has the value 0. The size of the array is not initialized by default. This means that if the `den_coeff` parameter has more than one coefficient, the `int_ic` parameter will still only have the single entry, 0, if used in its default mode. The mismatch in array sizes will cause errors when trying to run the simulation. If you intend to use the default value for `int_ic`, you must enter this value the required number of times, such that the number of entries match the number of coefficient entries in `den_coeff`. For example, if `den_coeff` had the entries:

```
1.9087 1.4325 0.28783
```

and you wished to use the default value (0) for `int_ic`, then you would need to enter the following for the `int_ic` parameter value:

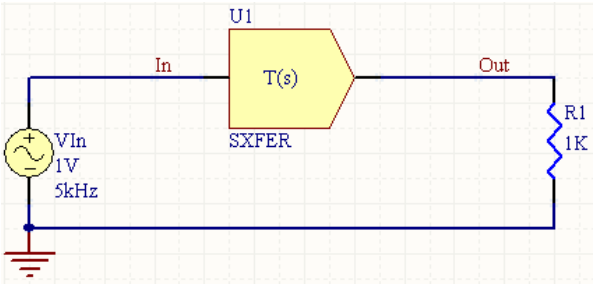
```
0 0 0
```

The provision of the `denormalized_freq` parameter allows you the freedom to either:

- specify the transfer function for a normalized (1 rad/s) filter and then enter the frequency of interest - effectively scaling the filter after the normalized coefficients have been defined. The frequency must be entered in radians/second.
- specify the transfer function and related coefficients directly for the frequency of interest. In this case, the `denormalization_freq` parameter can be left blank as the default value of 1 rad/s will be used.

Truncation error checking is an inherent part of the model. If truncation errors become excessive, the model uses smaller time increments between simulation data points, therefore providing for a more accurate simulation.

Examples



Consider the s-domain transfer function in the above image, with the following characteristics:

- Pin1 (in) is connected to net `In`
- Pin2 (out) is connected to net `Out`
- Designator is `U1`
- num_coeff = `1`
- den_coeff = `1 0.937 1.689 0.974 0.581 0.123`
- int_ic = `0 0 0 0 0 0`
- denormalized_freq = `18849.5559` rads/s (3kHz)
- All other model parameters are left at their inherent default values.

The transfer function represented by the model is that of a normalized 5th order Chebychev lowpass filter, with passband ripple of 1dB. The value entered in the `denormalized_freq` parameter will move the corner frequency to 3kHz (from the normalized 1 rad/s, or 159mHz).

The normalized transfer function for the filter is:

$$\frac{1}{}$$

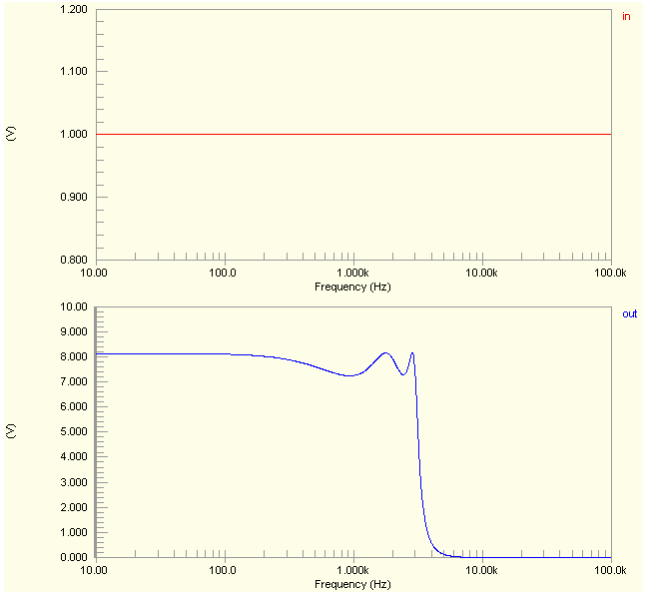
$$G(s) = \frac{}{1s^5 + 0.937s^4 + 1.689s^3 + 0.974s^2 + 0.581s + 0.123}$$

The entry in the SPICE netlist would be:

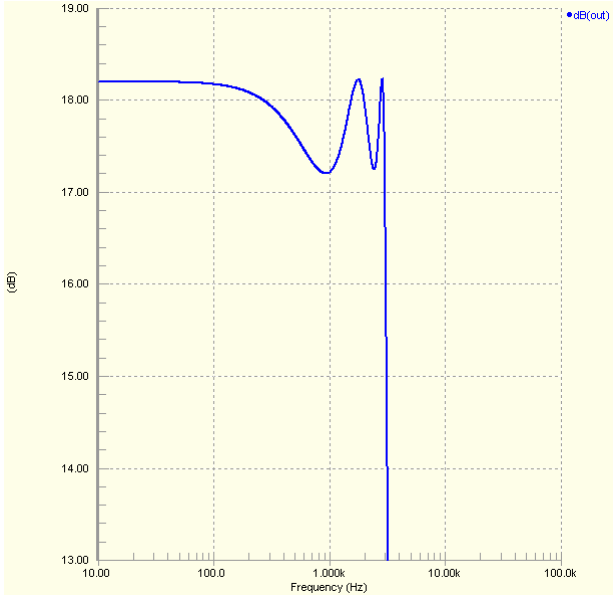
```
*Schematic Netlist:
AU1 IN OUT AU1SXFER
.MODEL AU1SXFER s_xfer (  num_coeff=[1] den_coeff=[1 0.937 1.689 0.974 0.581
+ 0.123] int_ic=[0 0 0 0 0 0] denormalized_freq=18849.5559)
```

The effect of the function can be seen in the resultant waveforms obtained by running an AC Small Signal analysis of the circuit. In this example, the following analysis parameters on the **AC Small Signal Analysis** page of the *Analyses Setup* dialog have been used:

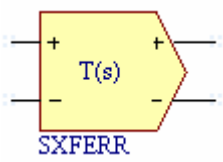
- **Start Frequency** – set to `10.00`
- **Stop Frequency** – set to `100.0k`
- **Sweep Type** – set to `Decade`
- **Test Points** – set to `500`.



By plotting the magnitude response in dBs, the corner frequency can be seen more clearly:



S-Domain Transfer Function (Differential I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

S_XFER

SPICE Netlist Template Format

```
@DESIGNATOR %%vd(%1,%2) %%vd(%3,%4) @"DESIGNATOR"SXFER
.MODEL @"DESIGNATOR"SXFER s_xfer (?in_offset|in_offset=@in_offset| ?gain|gain=@gain|
num_coeff=[@num_coeff] den_coeff=[@den_coeff] ?int_ic|int_ic=[@int_ic]|
?denormalized_freq|denormalized_freq=@denormalized_freq|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

in_offset	-	input offset (Default = 0).
gain	-	gain (Default = 1).
num_coeff	-	numerator polynomial coefficients. Enter a list of values, using spaces as separators. At least one value must be entered for the array.
den_coeff	-	denominator polynomial coefficients. Enter a list of values, using spaces as separators. At least one value must be entered for the array.
int_ic	-	integrator stage initial conditions. (Default = 0).
denormalized_freq	-	denormalized corner frequency (in radians). This allows you to specify the coefficients for a normalized filter, where the frequency of interest is 1 rad/s, and then move the corner frequency to the one of interest (denormalizing the transfer function). (Default = 1).

Notes

This model provides a single input, single output transfer function in the Laplace transform variable, **s**. This function enables you to modulate the frequency-domain characteristics of a signal.

The s-domain transfer function you define must adhere to the following two restrictions:

- The degree of the numerator polynomial cannot exceed that of the denominator polynomial.
- All polynomial coefficients must be stated explicitly, even if a coefficient is zero.

The model takes the differential input signal, applies any offset and gain specified by the `in_offset` and `gain` parameters and then multiplies the result by the transfer function determined by the polynomial coefficient entered in the respective `num_coeff` and `den_coeff` parameters.

When specifying the coefficients for numerator and denominator, the highest powered term coefficient must be entered first, followed by those coefficients for subsequent decreasing power terms.

There are no limits on the internal signal values, or on the output of the transfer function. Care should therefore be taken when specifying coefficients and gain, so that excessively large output values do not result.

In AC Small Signal analysis, the output of the function is equal to the real and imaginary components of the total s-domain transfer function at each frequency of interest.

The `int_ic` parameter is an array that must be the same size as the array of values specified for the `den_coeff` parameter. For example, if there are three coefficient entries defined in the `den_coeff` parameter, then the `int_ic` parameter must also have three entries, using spaces as separators. By default, this parameter has the value 0. The size of the array is not initialized by default. This means that if the `den_coeff` parameter has more than one coefficient, the `int_ic` parameter will still only have the single entry, 0, if used in its default mode. The mismatch in array sizes will cause errors when trying to run the simulation. If you intend to use the default value for `int_ic`, you must enter this value the required number of times, such that the number of entries match the number of coefficient entries in `den_coeff`. For example, if `den_coeff` had the entries:

```
1.9087 1.4325 0.28783
```

and you wished to use the default value (0) for `int_ic`, then you would need to enter the following for the `int_ic` parameter value:

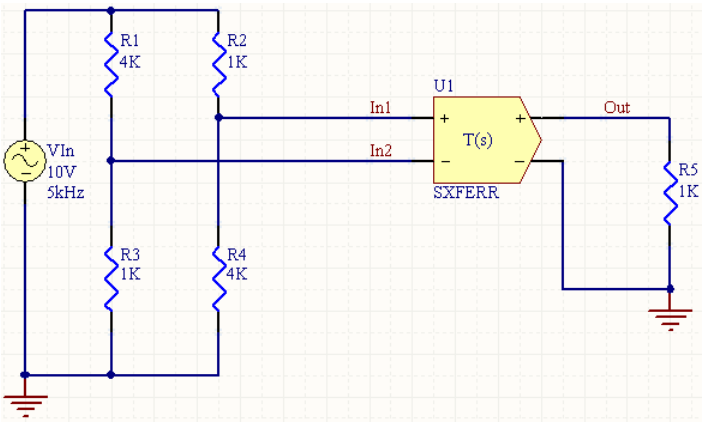
```
0 0 0
```

The provision of the `denormalized_freq` parameter allows you the freedom to either:

- specify the transfer function for a normalized (1 rad/s) filter and then enter the frequency of interest - effectively scaling the filter after the normalized coefficients have been defined. The frequency must be entered in radians/second.
- specify the transfer function and related coefficients directly for the frequency of interest. In this case, the `denormalization_freq` parameter can be left blank as the default value of 1 rad/s will be used.

Truncation error checking is an inherent part of the model. If truncation errors become excessive, the model uses smaller time increments between simulation data points, therefore providing for a more accurate simulation.

Examples



Consider the s-domain transfer function in the above image, with the following characteristics:

- Pin1 (positive input) is connected to net `In1`
- Pin2 (negative input) is connected to net `In2`
- Pin3 (positive output) is connected to net `Out`
- Pin4 (negative output) is connected to net `GND`
- Designator is `U1`
- num_coeff = `1`
- den_coeff = `1 2.6131 3.4142 2.6131 1`
- int_ic = `0 0 0 0 0`
- denormalized_freq = `18849.5559` rads/s (3kHz)
- All other model parameters are left at their inherent default values.

The transfer function represented by the model is that of a normalized 4th order Butterworth lowpass filter. The value entered in the `denormalized_freq` parameter will move the corner frequency to 3kHz (from the normalized 1 rad/s, or 159mHz).

The normalized transfer function for the filter is:

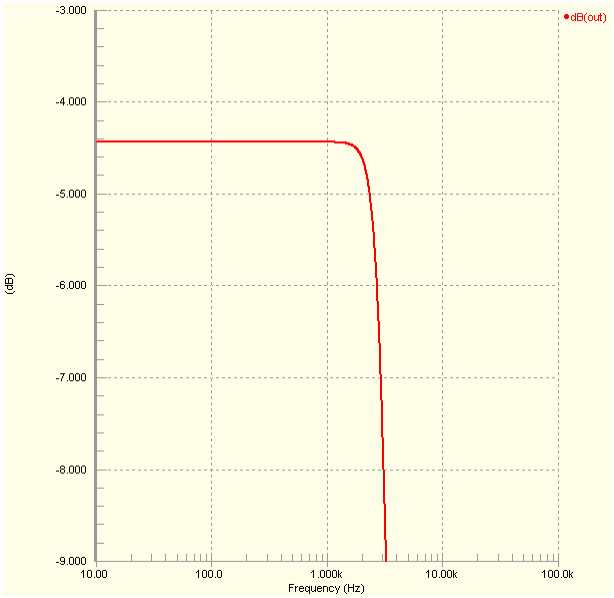
$$\frac{1}{}$$

$$G(s) = \frac{}{1s^4 + 2.6131s^3 + 3.4142s^2 + 2.6131s + 1}$$

The entry in the SPICE netlist would be:

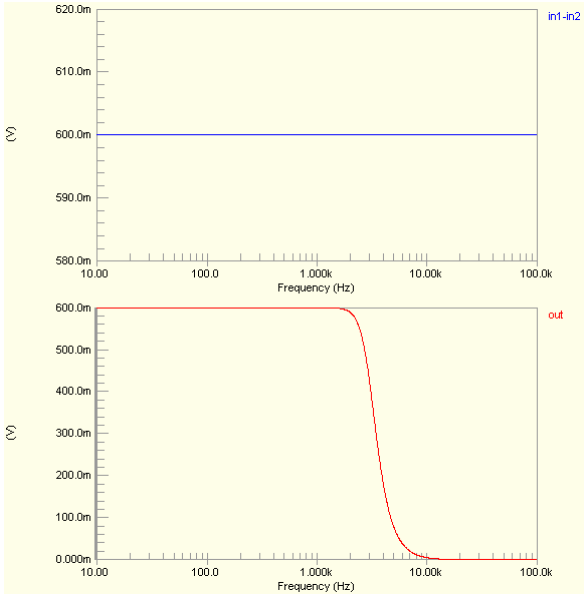
```
*Schematic Netlist:
AU1 %vd(IN1,IN2) %vd(OUT,0) AU1SXFER
.MODEL AU1SXFER s_xfer (  num_coeff=[1] den_coeff=[1 2.6131 3.4142 2.6131 1]
+ int_ic=[0 0 0 0 0] denormalized_freq=18849.5559)
```

The effect of the function can be seen in the resultant waveforms obtained by running an AC Small Signal analysis of the circuit.



In this example, the following analysis parameters on the **AC Small Signal Analysis** page of the *Analyses Setup* dialog have been used:

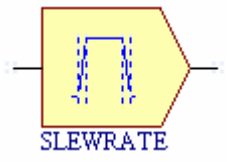
- **Start Frequency** – set to `10.00`
- **Stop Frequency** – set to `100.0k`
- **Sweep Type** – set to `Decade`
- **Test Points** – set to `500`.



By plotting the magnitude response in dBs, the corner frequency can be seen more clearly.

Slew Rate

Slew Rate (Single-Ended I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

SLEW

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @"DESIGNATOR"SLEW
.MODEL @"DESIGNATOR"SLEW slew (?rise_slope|rise_slope=@rise_slope|
?fall_slope|fall_slope=@fall_slope|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Rise_Slope - maximum rising slope value (Default = 1.0e9).

Fall_Slope - maximum falling slope value (Default = 1.0e9).

Notes

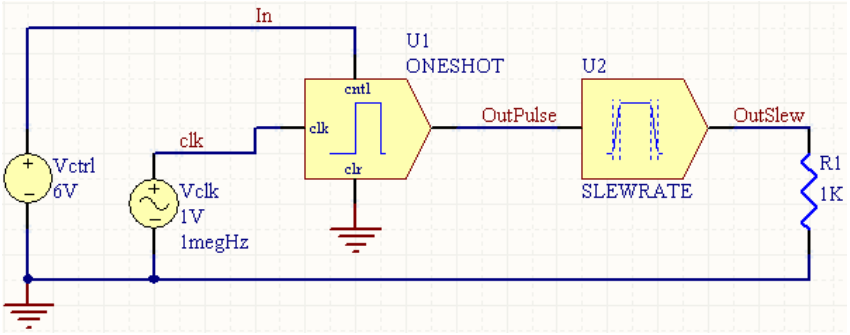
This model provides a simple slew rate feature that limits the absolute rising and falling slope of the output, with respect to time, to a defined value.

The values for the `Rise_Slope` and `Fall_Slope` parameters are specified in units of Vs^{-1} or As^{-1}. For example, to enter a slew rate of 0.05V/us, the entry would be 0.5e7.

The function will raise or lower the output value until the difference between the input and output is zero. It will then follow the input until the rise or fall slope limits are again exceeded.

The input signal can be either a single-ended current or single-ended voltage signal.

Examples



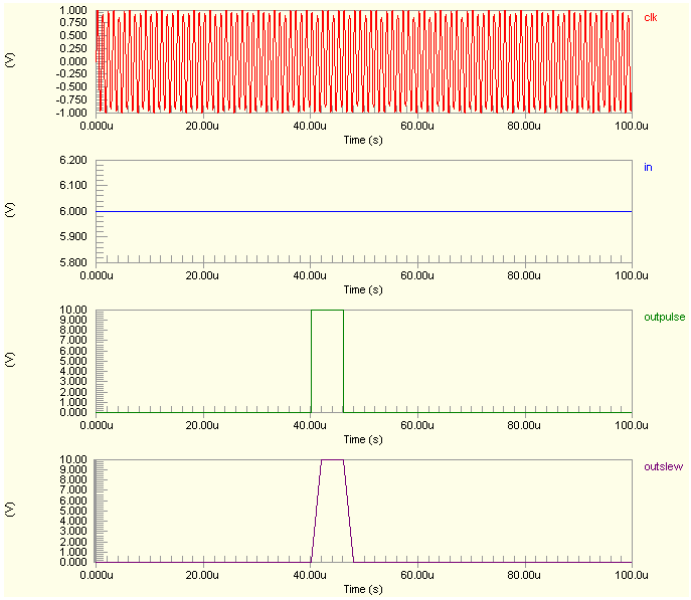
Consider the slew rate function in the above image, with the following characteristics:

- Pin1 (in) is connected to net `OutPulse`
- Pin2 (out) is connected to net `OutSlew`
- Designator is `U2`
- Rise_Slope = `0.5e7`
- Fall_Slope = `0.5e7`

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU2 OUTPULSE OUTSLEW AU2SLEW
.MODEL AU2SLEW slew (rise_slope=0.5e7 fall_slope=0.5e7)
```

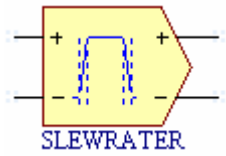
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `100.0u`
- **Transient Step Time** – set to `20.00n`
- **Transient Max Step Time** – set to `20.00n`.

Slew Rate (Differential I/O)



Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

Model Name

SLEW

SPICE Netlist Template Format

```
@DESIGNATOR %%vd(%1,%2) %%vd(%3,%4) @"DESIGNATOR"SLEW
.MODEL @"DESIGNATOR"SLEW slew (?rise_slope|rise_slope=@rise_slope|
?fall_slope|fall_slope=@fall_slope|)
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Rise_Slope - maximum rising slope value (Default = 1.0e9).

Fall_Slope - maximum falling slope value (Default = 1.0e9).

Notes

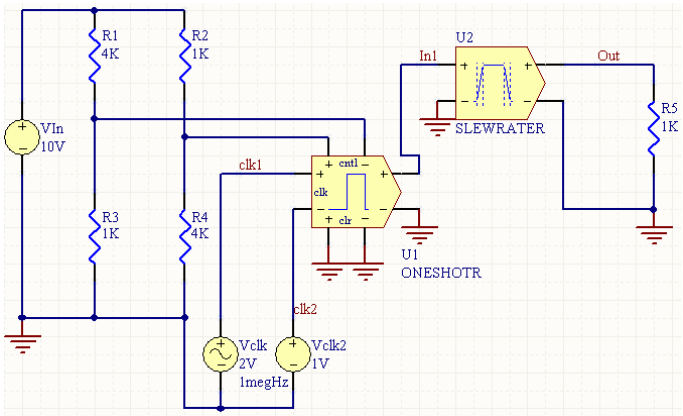
This model provides a simple slew rate feature that limits the absolute rising and falling slope of the output, with respect to time, to a defined value.

The values for the `Rise_Slope` and `Fall_Slope` parameters are specified in units of Vs^{-1} or As^{-1}. For example, to enter a slew rate of 0.05V/us, the entry would be 0.5e7.

The function will raise or lower the output value until the difference between the input and output is zero. It will then follow the input until the rise or fall slope limits are again exceeded.

The input signal can be either a differential current or differential voltage signal.

Examples



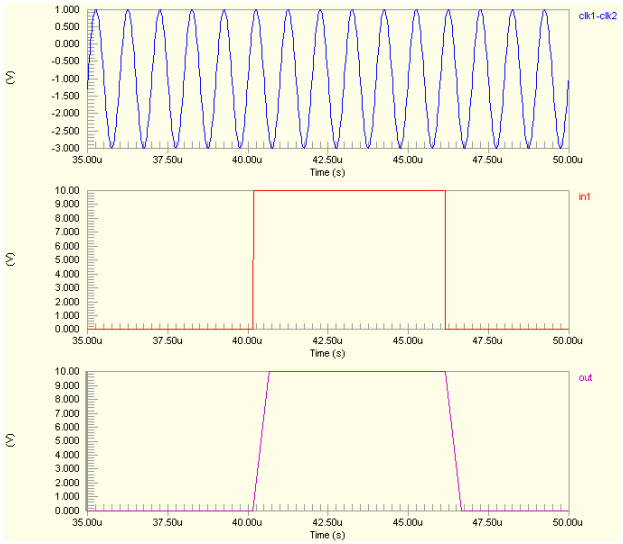
Consider the slew rate function in the above image, with the following characteristics:

- Pin1 (positive input) is connected to net `In1`
- Pin2 (negative input) is connected to net `GND`
- Pin3 (positive output) is connected to net `Out`
- Pin4 (negative output) is connected to net `GND`
- Designator is `U2`
- Rise_Slope = `2e7`
- Fall_Slope = `2e7`.

The entry in the SPICE netlist would be:

```
*Schematic Netlist:
AU2 %vd(IN1,0) %vd(OUT,0) AU2SLEW
.MODEL AU2SLEW slew (rise_slope=2e7 fall_slope=2e7)
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

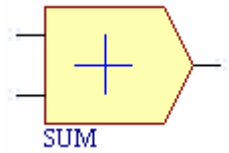


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `100.0u`
- **Transient Step Time** – set to `20.00n`
- **Transient Max Step Time** – set to `20.00n`.

Summer

Summer (Single-Ended I/O)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

SUM

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 @MODEL #|PARAMS:| ?X_OFFSET|X_OFFSET=@X_OFFSET|
?Y_OFFSET|Y_OFFSET=@Y_OFFSET| ?X_GAIN|X_GAIN=@X_GAIN| ?Y_GAIN|Y_GAIN=@Y_GAIN|
?OUT_GAIN|OUT_GAIN=@OUT_GAIN| ?OUT_OFFSET|OUT_OFFSET=@OUT_OFFSET|
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

X_Offset - X input offset (Default = 0).

Y_Offset - Y input offset (Default = 0).

X_Gain - X input gain (Default = 1).

Y_Gain - Y input gain (Default = 1).

Out_Gain - output gain (Default = 1).

Out_Offset - output offset (Default = 0).

Notes

This is a two-input summer with offset and gain adjustment available on both inputs and output. It takes the inputs and processes them to obtain the output result as follows:

- The inputs are offset, in accordance with the values specified for the `X_Offset` and `Y_Offset` parameters.
- The offset signals are then multiplied by the values for gain specified in the respective `X_Gain` and `Y_Gain` parameters.
- The resulting values are summed
- The result is then multiplied by the value specified for the `Out_Gain` parameter.
- The output result is then offset in accordance with the value specified for the `Out_Offset` parameter.

The process can be expressed mathematically as follows:

Output =((((X + X_Offset) * X_Gain) + ((Y + Y_Offset) * Y_Gain)) * Out_Gain) + Out_Offset

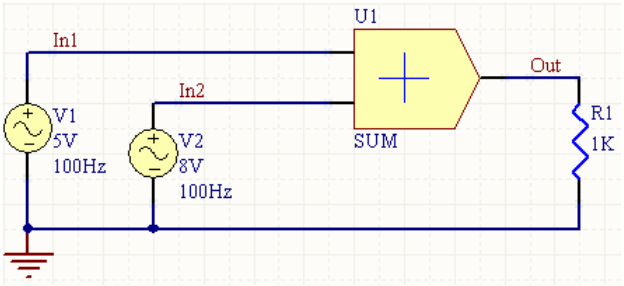
The input signals can be either single-ended current or single-ended voltage signals.

The built-in XSpice summer function can take two or more inputs, with no upper limit on the number of inputs considered. This particular 2-input version is defined using the hierarchical sub-circuit syntax. Within the sub-circuit definition, the XSpice summer model is called and the parameters of the sub-circuit file parsed to this model.

Entering a value for a parameter on the **Parameters** tab of the *Sim Model* dialog will override its specified value in the sub-circuit file.

To check the default values of the supplied 2-input summer, open the appropriate sub-circuit (`.ckt`) file. You can view the content of this file for the model specified on the **Model Kind** tab of the *Sim Model* dialog, by clicking on the **Model File** tab at the bottom of the dialog. The default parameter values are listed in the `.SUBCKT` line.

Examples



Consider the summer in the above image, with the following characteristics:

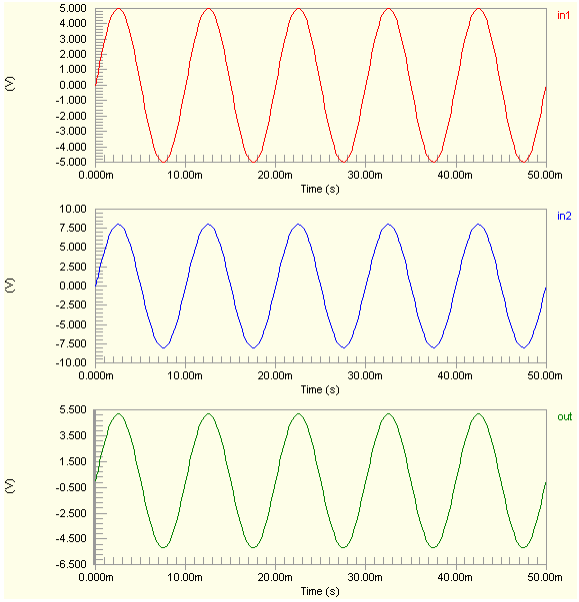
- Pin1 (in_a) is connected to net `In1`
- Pin2 (in_b) is connected to net `In2`
- Pin3 (out) is connected to net `Out`
- Designator is `U1`
- X_Offset = `2V` (defined on the **Parameters** tab)
- X_Gain = `4` (defined on the **Parameters** tab)
- Y_Offset = `4V` (defined on the **Parameters** tab)
- Y_Gain = `4` (defined on the **Parameters** tab)
- Out_Gain = `0.1` (defined on the **Parameters** tab)
- Out_Offset = `-2.4V` (defined on the **Parameters** tab).

The entries in the SPICE netlist would be:

```
*Schematic Netlist:
```

```
XU1 IN1 IN2 OUT SUM PARAMS: x_offset=2V y_offset=4V x_gain=4 y_gain=4
+ out_gain=0.1 out_offset=-2.4V
.
.
*Models and Subcircuit:
.SUBCKT SUM 1 2 3 PARAMS: x_offset=0.0 y_offset=0.0 x_gain=1.0 y_gain=1.0
+ out_gain=1.0 out_offset=0.0
A1 [1 2] 3 sum1
.model sum1 summer(in_offset=[{x_offset} {y_offset}]  in_gain=[{x_gain} {y_gain}]
+ out_gain={out_gain}  out_offset={out_offset})
.ENDS SUM
```

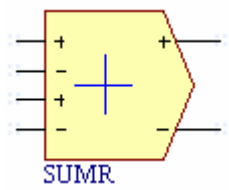
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Summer (Differential I/O)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

SUMR

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 %5 %6 @MODEL #|PARAMS:| ?x_offset|x_offset=@x_offset|
?y_offset|y_offset=@y_offset| ?x_gain|x_gain=@x_gain| ?y_gain|y_gain=@y_gain|
?out_gain|out_gain=@out_gain| ?out_offset|out_offset=@out_offset|
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

X_Offset - X input offset (Default = 0).

Y_Offset - Y input offset (Default = 0).

X_Gain - X input gain (Default = 1).

Y_Gain - Y input gain (Default = 1).

Out_Gain - output gain (Default = 1).

Out_Offset - output offset (Default = 0).

Notes

This is a two-input summer with offset and gain adjustment available on both inputs and output. It takes the inputs and processes them to obtain the output result as follows:

- The inputs are offset, in accordance with the values specified for the `X_Offset` and `Y_Offset` parameters.
- The offset signals are then multiplied by the values for gain specified in the respective `X_Gain` and `Y_Gain` parameters.
- The resulting values are summed
- The result is then multiplied by the value specified for the `Out_Gain` parameter.
- The output result is then offset in accordance with the value specified for the `Out_Offset` parameter.

The process can be expressed mathematically as follows:

Output =((((X + X_Offset) * X_Gain) + ((Y + Y_Offset) * Y_Gain)) * Out_Gain) + Out_Offset

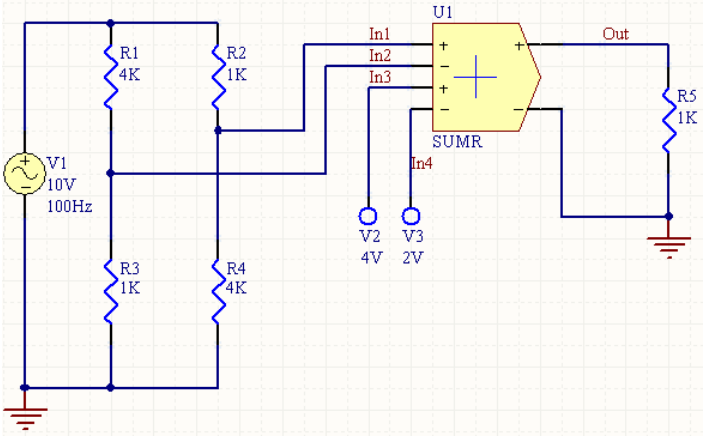
The input signals can be either differential current or differential voltage signals.

The built-in XSpice summer function can take two or more inputs, with no upper limit on the number of inputs considered. This particular 2-input version is defined using the hierarchical sub-circuit syntax. Within the sub-circuit definition, the XSpice summer model is called and the parameters of the sub-circuit file parsed to this model.

Entering a value for a parameter on the **Parameters** tab of the *Sim Model* dialog will override its specified value in the sub-circuit file.

To check the default values of the supplied 2-input summer, open the appropriate sub-circuit (`.ckt`) file. You can view the content of this file for the model specified on the **Model Kind** tab of the *Sim Model* dialog, by clicking on the **Model File** tab at the bottom of the dialog. The default parameter values are listed in the `.SUBCKT` line.

Examples



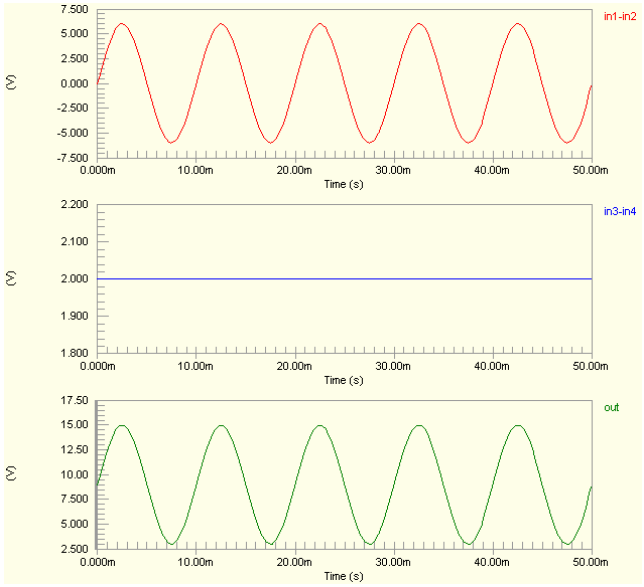
Consider the summer in the above image, with the following characteristics:

- Pin1 (positive a input) is connected to net `In1`
- Pin2 (negative a input) is connected to net `In2`
- Pin3 (positive b input) is connected to net `In3`
- Pin4 (negative b input) is connected to net `In4`
- Pin5 (positive output) is connected to net `Out`
- Pin6 (negative output) is connected to net `GND`
- Designator is `U1`
- X_Gain = `0.5` (defined on the **Parameters** tab)
- Y_Gain = `2.25` (defined on the **Parameters** tab)
- Out_Gain = `2` (defined on the **Parameters** tab)
- All other model parameters are left at their inherent default values.

The entries in the SPICE netlist would be:

```
*Schematic Netlist:
XU1 IN1 IN2 IN3 IN4 OUT 0 SUMR PARAMS:   x_gain=0.5 y_gain=2.25 out_gain=2
.
.
*Models and Subcircuit:
.SUBCKT SUMR 1   2   3   4   5   6 PARAMS: x_offset=0.0 y_offset=0.0 x_gain=1.0
+ y_gain=1.0 out_gain=1.0 out_offset=0.0
A1 [%vd(1,2) %vd(3,4)] %vd(5,6) sum1
.model sum1 summer(in_offset=[{x_offset} {y_offset}] in_gain=[{x_gain} {y_gain}]
+ out_gain={out_gain} out_offset={out_offset})
.ENDS SUMR
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Sub-circuit based models

These are models for more complex devices, such as operational amplifiers, timers, crystals, etc, that have been described using the hierarchical sub-circuit syntax.

A sub-circuit consists of SPICE elements that are defined and referenced in a fashion similar to device models. There is no limit on the size or complexity of sub-circuits and sub-circuits can call other sub-circuits. Each sub-circuit is defined in a sub-circuit file (`*.ckt`).

The following sub-circuit based device model examples are covered in this section:

- *Crystal*
- *Frequency to Voltage Converter*
- *Fuse*
- *Relay*
- *Transformer (Equivalent Circuit Model)*
- *Voltage Controlled Sine Wave Oscillator*
- *Voltage Controlled Square Wave Oscillator*
- *Voltage Controlled Triangle Wave Oscillator*.

Notes

The SPICE prefix for theses models is X.

Many of the component libraries (`*.IntLib`) that come with the installation, feature simulation-ready devices. These devices have the necessary model or sub-circuit file included and linked to the schematic component. These are pure SPICE models for maximum compatibility with analog simulators.

Crystal

Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @MODEL #|PARAMS:| ?FREQ|FREQ=@FREQ| ?RS|RS=@RS| ?C|C=@C| ?Q|Q=@Q|
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

FREQ - the nominal output frequency of the crystal (in Hertz).

RS - the resistance exhibited by the crystal at the series resonant frequency (in Ohms).

C - shunt capacitance (in Farads). This value is the combination of the capacitance due to the electrodes on the crystal plate and stray capacitances arising from the crystal holder/enclosure.

Q - the quality factor of the equivalent electrical circuit model for the crystal. This essentially is the ratio of energy stored to energy dissipated for the circuit and can be further defined as the ratio of the reactance (series motional inductance and capacitance) to the series resistance at the resonant frequency.

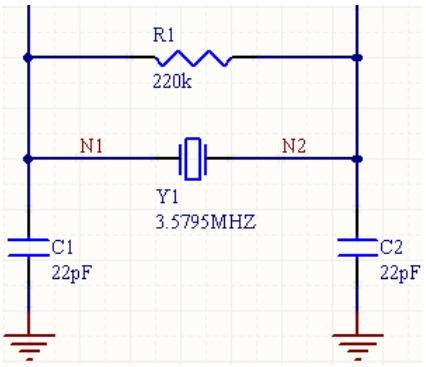
Notes

A crystal is not one of the built-in SPICE engine models. It is a complex device and, as such, is defined using the hierarchical sub-circuit syntax.

All of the parameters will normally have a default value assigned. The default should be applicable to most simulations. Generally you do not need to change this value. Entering a value for a parameter on the **Parameters** tab of the *Sim Model* dialog will override its specified value in the sub-circuit file.

To check the default values of a crystal, open the appropriate sub-circuit (`.ckt`) file. You can view the content of this file for the model specified on the **Model Kind** tab of the *Sim Model* dialog, by clicking on the **Model File** tab at the bottom of the dialog. The default parameter values are listed in the `.SUBCKT` line.

Examples



Consider the crystal in the above image, with the following characteristics:

- Pin1 is connected to net `N1`
- Pin2 is connected to net `N2`
- Designator is `Y1`
- The linked simulation sub-circuit file is `3.5795MHz.ckt` with the following content:

```
*Crystal Subcircuit Parameters
*FREQ = Fundamental frequency
*RS   = Series resistance
*C    = Parallel capacitance
*Q    = Quality Factor
*CTS Color Burst alias:XCRYSTAL {FREQ=3.58E6 RS=160 C=1.8E-11} pkg:HC49
.SUBCKT 3.5795MHZ 1 2 PARAMS: FREQ=3.58E6 RS=160 C=1.8E-11 Q=1000
LX 1 3 {((Q*RS)/(6.2831852*FREQ))} IC=0.5M
CX 3 4 {(1/(Q*6.2831852*FREQ*RS))}
C0 1 2 {C}
RS 4 2 {RS}
.ENDS
```

If no values are entered for the parameters in the *Sim Model* dialog, the entries in the SPICE netlist would be:

```
*Schematic Netlist:
XY1 N1 N2 3.5795MHZ
.
.
*Models and Subcircuit:
.SUBCKT 3.5795MHZ 1 2 PARAMS: FREQ=3.58E6 RS=160 C=1.8E-11 Q=1000
LX 1 3 {((Q*RS)/(6.2831852*FREQ))} IC=0.5M
CX 3 4 {(1/(Q*6.2831852*FREQ*RS))}
C0 1 2 {C}
RS 4 2 {RS}
```

```
.ENDS
```

If the following overriding parameter values were specified on the **Parameters** tab of the *Sim Model* dialog:

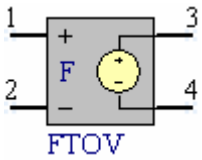
- **FREQ** = `10MEGHz`
- **Q** = `10000`

then the entries in the SPICE netlist would be:

```
*Schematic Netlist:
XY1 N1 N2 3.5795MHZ PARAMS: FREQ=10MEGHz   Q=10000
.
.
*Models and Subcircuit:
.SUBCKT 3.5795MHZ 1 2 PARAMS: FREQ=3.58E6 RS=160 C=1.8E-11 Q=1000
LX 1 3 {((Q*RS)/(6.2831852*FREQ))} IC=0.5M
CX 3 4 {(1/(Q*6.2831852*FREQ*RS))}
C0 1 2 {C}
RS 4 2 {RS}
.ENDS
```

In this case, the Netlister will evaluate the formulae in the sub-circuit definition using the values for `FREQ` and `Q` (from the *Sim Model* dialog) and the default values for `RS` and `C`, as defined in the `3.5795MHZ.ckt` file.

Frequency to Voltage Converter



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL #|PARAMS:| ?VIL|VIL=@VIL| ?VIH|VIH=@VIH|
?CYCLES|CYCLES=@CYCLES|
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

VIL - low level input threshold (in Volts).

VIH - high level input threshold (in Volts).

CYCLES - cycles per volt output.

Notes

The output is a voltage, the level of which is a linear function of the input frequency.

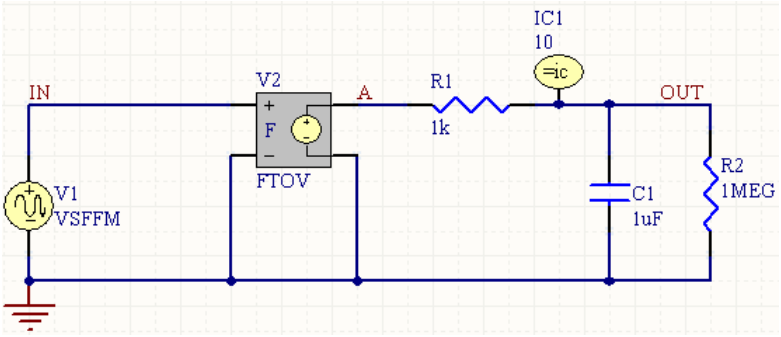
The frequency to voltage converter is not one of the built-in SPICE engine models. It is a complex device and, as such, is defined using the hierarchical sub-circuit syntax.

All of the parameters will normally have a default value assigned. The default should be applicable to most simulations. Generally you do not need to change this value. Entering a value for a parameter on the **Parameters** tab of the *Sim Model* dialog will override its specified value in the sub-circuit file.

To check the default values of the model, open the associated sub-circuit (`.ckt`) file. You can view the content of this file for the model specified on the **Model Kind** tab of the *Sim Model* dialog, by clicking on the **Model File** tab at the bottom of the dialog. The default parameter values are listed in the `.SUBCKT` line.

The simulation-ready frequency to voltage converter component (`FTOV`) can be found in the Simulation Special Function integrated library (`\Library\Simulation\Simulation Special Function.IntLib`).

Examples



Consider the frequency to voltage converter in the above image, with the following characteristics:

- Pin1 (positive controlling node) is connected to net `IN`
- Pin2 (negative controlling node) is connected to net `GND`
- Pin3 (positive output node) is connected to net `A`
- Pin4 (negative output node) is connected to net `GND`
- Designator is `V2`
- The linked simulation sub-circuit file is `FTOV.ckt`, with the following content:

```
*Frequency To Voltage Converter
*VIL    = Low level input threshold
*VIH    = High level input threshold
*CYCLES = Cycles per volt output
*Generic frequency to voltage converter
*Connections:
*            NC+
*            | NC-
*            | | N+
*            | | | N-
*            | | | |
.SUBCKT FTOV 1 2 3 4 PARAMS: VIL=1 VIH=2 CYCLES=1k
A2 [1 2] [10 20] adc_mod
A2 [10 20] [40] fcvs_mod
A3 [40] [5] dav_mod
B1 3 4 V=(v(5)/{CYCLES})
.model adc_mod xadc
.model dav_mod xdav
.model fcvs_mod xsimcode(file="{MODEL_PATH}fcvs.scb" func=fcvs VIL={VIL} VIH={VIH})
.ENDS FTOV
```

- Values of `0.1` and `0.2` are entered respectively for VIL and VIH on the **Parameters** tab of the *Sim Model* dialog.

The entries in the SPICE netlist would be:

```
*Schematic Netlist:
XV2 IN 0 A 0 FTOV PARAMS: VIL=.1 VIH=.2
.
.
*Models and Subcircuit:
.SUBCKT FTOV 1 2 3 4 PARAMS: VIL=1 VIH=2 CYCLES=1k
A2 [1 2] [10 20] adc_mod
A2 [10 20] [40] fcvs_mod
A3 [40] [5] dav_mod
B1 3 4 V=(v(5)/{CYCLES})
.model adc_mod xadc
.model dav_mod xdav
.model fcvs_mod
+ xsimcode(file="C:\Program Files\Altium Designer\Library\Sim\fcvs.scb" func=fcvs VIL={VIL}
VIH={VIH})
.ENDS FTOV
```

The Netlister will evaluate the formulae in the sub-circuit definition using the overriding values for the parameters `VIL` and `VIH` (as defined in the *Sim Model* dialog) and the default value for the parameter `CYCLES`, as defined in the `FTOV.ckt` file.

Fuse

Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @MODEL #|PARAMS:| ?RESISTANCE|RESISTANCE=@RESISTANCE|
?CURRENT|CURRENT=@CURRENT|
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

RESISTANCE - series fuse resistance (in Ohms).

CURRENT - fuse current at rupture (in Amps).

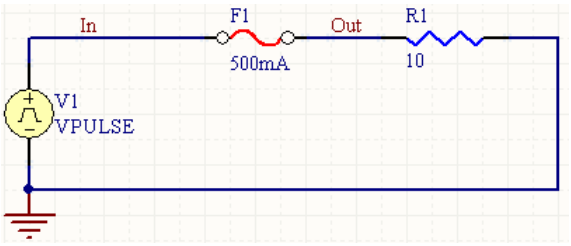
Notes

A fuse is not one of the built-in SPICE engine models. It is a complex device and, as such, is defined using the hierarchical sub-circuit syntax.

All of the parameters will normally have a default value assigned. The default should be applicable to most simulations. Generally you do not need to change this value. Entering a value for a parameter on the **Parameters** tab of the *Sim Model* dialog will override its specified value in the sub-circuit file.

To check the default values of a fuse, open the appropriate sub-circuit (`.ckt`) file. You can view the content of this file for the model specified on the **Model Kind** tab of the *Sim Model* dialog, by clicking on the **Model File** tab at the bottom of the dialog. The default parameter values are listed in the `.SUBCKT` line.

Examples



Consider the fuse in the above image, with the following characteristics:

- Pin1 is connected to net `In`
- Pin2 is connected to net `Out`
- Designator is `F1`
- The linked simulation sub-circuit file is `FUSE.ckt`, with the following content:

```
*FUSE:Fuse Subcircuit Parameters
*CURRENT    = Fuse current
*RESISTANCE = Inernal resistance
.SUBCKT FUSE 1 2 PARAMS: CURRENT=1 RESISTANCE=1m
SW1 1 2 3 0 SMOD OFF
BNLV 3 0 V=(abs(v(1,2)))
.MODEL SMOD SW (VT={(CURRENT*RESISTANCE)} RON=1g ROFF={RESISTANCE})
.ENDS FUSE
```

- CURRENT = `500mA` - set on the **Parameters** tab of the *Sim Model* dialog.

The entries in the SPICE netlist would be:

```
*Schematic Netlist:
XF1 IN OUT FUSE PARAMS:  CURRENT=500mA
.
.
*Models and Subcircuit:
.SUBCKT FUSE 1 2 PARAMS: CURRENT=1 RESISTANCE=1m
SW1 1 2 3 0 SMOD OFF
BNLV 3 0 V=(abs(v(1,2)))
.MODEL SMOD SW (VT={(CURRENT*RESISTANCE)} RON=1g ROFF={RESISTANCE})
.ENDS FUSE
```

The Netlister will evaluate the formulae in the sub-circuit definition using the value for the `CURRENT` parameter specified in the *Sim Model* dialog (which overrides the default) and the default value for the `RESISTANCE` parameter (1 mOhm), as defined in the `FUSE.ckt` file.

Relay

Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 %5 @MODEL #|PARAMS:| ?PULLIN|PULLIN=@PULLIN|
?DROPOFF|DROPOFF=@DROPOFF| ?CONTACT|CONTACT=@CONTACT| ?RESISTANCE|RESISTANCE=@RESISTANCE|
?INDUCTANCE|INDUCTANCE=@INDUCTANCE|
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Pullin	-	contact pull in voltage (in Volts).
Dropoff	-	contact drop off voltage (in Volts).
Contact	-	contact resistance (in Ohms).
Resistance	-	coil resistance (in Ohms).
Inductance	-	coil inductance (in Henrys).

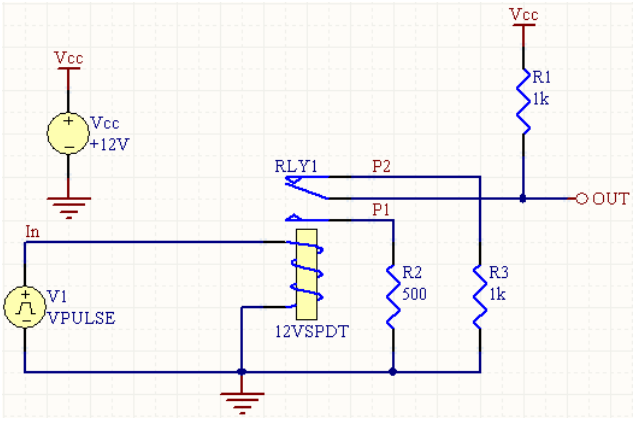
Notes

A relay is not one of the built-in SPICE engine models. It is a complex device and, as such, is defined using the hierarchical sub-circuit syntax.

All of the parameters will normally have a default value assigned. The default should be applicable to most simulations. Generally you do not need to change this value. Entering a value for a parameter on the **Parameters** tab of the *Sim Model* dialog will override its specified value in the sub-circuit file.

To check the default values of a relay, open the appropriate sub-circuit (`.ckt`) file. You can view the content of this file for the model specified on the **Model Kind** tab of the *Sim Model* dialog, by clicking on the **Model File** tab at the bottom of the dialog. The default parameter values are listed in the `.SUBCKT` line.

Examples



Consider the relay in the above image, with the following characteristics:

- Pin1 is connected to net `OUT`
- Pin2 is connected to net `P2`
- Pin3 is connected to net `P1`
- Pin4 is connected to net `In`
- Pin5 is connected to net `GND`
- Designator is `RLY1`
- The linked simulation sub-circuit file is `12VSPDT.ckt`, with the following content:

```
*Generic relay pins: COM NC NO T1 T2
*SPDT Relay Subcircuit Parameters
*PULLIN      = Pull in voltage
*DROPOFF     = Drop off voltage
```

```
*CONTACT    = Contact resistance
*RESISTANCE = Coil resistance
*INDUCTANCE = Coil Inductance
*AC/DC relay
.SUBCKT 12VSPDT 1 2 3 4 5
+ PARAMS: PULLIN=9.6 DROPOFF=0.1 CONTACT=1m RESISTANCE=1000 INDUCTANCE=10m
L1 4 6 {(INDUCTANCE/2)}
L2 5 7 {(INDUCTANCE/2)}
R1 6 7 {RESISTANCE}
BNO 8 0 V={PULLIN}-abs(v(6,7))
SW1 2 1 8 0 SWNC ON
BNC 9 0 V=abs(v(6,7))
SW2 3 1 9 0 SWNO OFF
.MODEL SWNC SW(VT={DROPOFF} RON={CONTACT} )
.MODEL SWNO SW(VT={(PULLIN*0.98)} RON={CONTACT} )
.ENDS SPDTRELAY
```

- Pullin = `8.4V` - set on the **Parameters** tab of the *Sim Model* dialog.

The entries in the SPICE netlist would be:

```
*Schematic Netlist:
XRLY1 OUT P2 P1 IN 0 12VSPDT PARAMS: Pullin=8.4V
.
.
*Models and Subcircuit:
.SUBCKT 12VSPDT 1 2 3 4 5 PARAMS: PULLIN=9.6 DROPOFF=0.1 CONTACT=1m
+ RESISTANCE=1000 INDUCTANCE=10m
L1 4 6 {(INDUCTANCE/2)}
L2 5 7 {(INDUCTANCE/2)}
R1 6 7 {RESISTANCE}
BNO 8 0 V={PULLIN}-abs(v(6,7))
SW1 2 1 8 0 SWNC ON
BNC 9 0 V=abs(v(6,7))
SW2 3 1 9 0 SWNO OFF
.MODEL SWNC SW(VT={DROPOFF} RON={CONTACT} )
.MODEL SWNO SW(VT={(PULLIN*0.98)} RON={CONTACT} )
.ENDS SPDTRELAY
```

The Netlister will evaluate the formulae in the sub-circuit definition using the value for the `Pullin` parameter specified in the *Sim Model* dialog (which overrides the default) and the default values for all other parameters, as defined in the `12VSPDT.ckt` file.

Transformer (Equivalent Circuit Model)

Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL #|PARAMS:| ?RATIO|RATIO=@RATIO| ?RP|RP=@RP| ?RS|RS=@RS|
?LEAK|LEAK=@LEAK| ?MAG|MAG=@MAG|
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog

RATIO - turns ratio (secondary/primary).

RP - resistance of primary winding (in Ohms).

RS - resistance of secondary winding (in Ohms).

LEAK - leakage inductance (in Henrys).

MAG - magnetizing inductance (in Henrys).

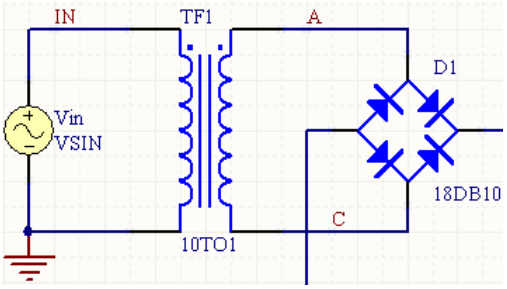
Notes

A transformer is not one of the built-in SPICE engine models. It is a complex device and, as such, is defined using the hierarchical sub-circuit syntax.

All of the parameters will normally have a default value assigned. The default should be applicable to most simulations. Generally you do not need to change this value. Entering a value for a parameter on the **Parameters** tab of the *Sim Model* dialog will override its specified value in the sub-circuit file.

To check the default values of a transformer, open the appropriate sub-circuit (`.ckt`) file. You can view the content of this file for the model specified on the **Model Kind** tab of the *Sim Model* dialog, by clicking on the **Model File** tab at the bottom of the dialog. The default parameter values are listed in the `.SUBCKT` line.

Examples



Consider the transformer in the above image, with the following characteristics:

- Pin1 (Pri+) is connected to net `IN`
- Pin2 (Pri-) is connected to net `GND`
- Pin3 (Sec+) is connected to `A`
- Pin4 (Sec-) is connected to net `C`
- Designator is `TF1`
- The linked simulation sub-circuit file is `10TO1.ckt`, with the following content:

```
*Transformer Subcircuit Parameters
*RATIO = Turns ratio= Secondary/Primary
*RP    = Primary DC resistance
*RS    = Secondary DC resistance
*LEAK  = Leakage inductance
*MAG   = Magnetizing inductance
*10:1 Transformer
*Connections:
*             Pri+
```

```
*              | Pri-
*              | | Sec+
*              | | | Sec-
*              | | | |
.SUBCKT 10TO1 1 2 3 4 PARAMS: RATIO=0.1 RP=0.1 RS=0.1 LEAK=1u MAG=1u
VISRC 9 4 DC 0V
FCTRL 6 2 VISRC {RATIO}
EVCVS 8 9 5 2 {RATIO}
RPRI  1 7 {RP}
RSEC  8 3 {RS}
LLEAK 7 5 {LEAK}
LMAGNET 6 5 {MAG}
.ENDS 10TO1
```

If no overriding parameters for the model are specified on the **Parameters** tab of the *Sim Model* dialog, then the default values listed in the sub-circuit definition will be used and the entries in the SPICE netlist would be:

```
*Schematic Netlist:
XTF1 IN 0 A C 10TO1
.
.
*Models and Subcircuit:
.SUBCKT 10TO1 1 2 3 4 PARAMS: RATIO=0.1 RP=0.1 RS=0.1 LEAK=1u MAG=1u
VISRC 9 4 DC 0V
FCTRL 6 2 VISRC {RATIO}
EVCVS 8 9 5 2 {RATIO}
RPRI  1 7 {RP}
RSEC  8 3 {RS}
LLEAK 7 5 {LEAK}
LMAGNET 6 5 {MAG}
.ENDS 10TO1
```

The Netlister will evaluate the formulae in the sub-circuit definition using the default parameter values, as defined in the `10TO1.ckt` file.

Voltage Controlled Sine Wave Oscillator

Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL #|PARAMS:| ?LOW|LOW=@LOW| ?HIGH|HIGH=@HIGH| ?C1|C1=@C1|
?F1|F1=@F1| ?C2|C2=@C2| ?F2|F2=@F2| ?C3|C3=@C3| ?F3|F3=@F3| ?C4|C4=@C4| ?F4|F4=@F4|
?C5|C5=@C5| ?F5|F5=@F5|
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Low - peak output low value (in Volts).

High - peak output high value (in Volts).

C1 - input control voltage point 1 (in Volts).

F1 - output frequency point 1 (in Hertz).

C2 - input control voltage point 2 (in Volts).

F2 - output frequency point 2 (in Hertz).

C3 - input control voltage point 3 (in Volts).

F3 - output frequency point 3 (in Hertz).

C4 - input control voltage point 4 (in Volts).

F4 - output frequency point 4 (in Hertz).

C5 - input control voltage point 5 (in Volts).

F5 - output frequency point 5 (in Hertz).

Notes

The parameters `C1`, `C2`,`...` and `F1`, `F2`,`...` define the voltage to frequency conversion function. The C values define input voltage levels and the F values set the respective output frequencies generated for these input levels. Linear interpolation is used to define input/output values between the set points.

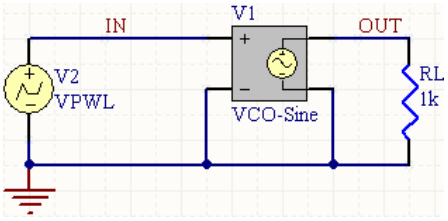
The voltage controlled sine wave oscillator is not one of the built-in SPICE engine models. It is a complex device and, as such, is defined using the hierarchical sub-circuit syntax.

All of the parameters will normally have a default value assigned. The default should be applicable to most simulations. Generally you do not need to change this value. Entering a value for a parameter on the **Parameters** tab of the *Sim Model* dialog will override its specified value in the sub-circuit file.

To check the default values of the model, open the associated sub-circuit (`.ckt`) file. You can view the content of this file for the model specified on the **Model Kind** tab of the *Sim Model* dialog, by clicking on the **Model File** tab at the bottom of the dialog. The default parameter values are listed in the `.SUBCKT` line.

The simulation-ready voltage controlled sine wave oscillator component (`VCO-Sine`) can be found in the Simulation Special Function integrated library (`\Library\Simulation\Simulation Special Function.IntLib`).

Examples



Consider the voltage controlled sine wave oscillator in the above image, with the following characteristics:

- Pin1 (positive controlling node) is connected to net `IN`
- Pin2 (negative controlling node) is connected to net `GND`

- Pin3 (positive output node) is connected to net `OUT`
- Pin4 (negative output node) is connected to net `GND`
- Designator is `V1`
- The linked simulation sub-circuit file is `SINEVCO.ckt`, with the following content:

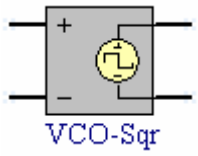
```
*Voltage Controlled Sine Wave Oscillator
* LOW  = Peak output low value
* HIGH = Peak output high value
* C1   = Input control voltage point 1
* C2   = Input control voltage point 2
* C3   = Input control voltage point 3
* C4   = Input control voltage point 4
* C5   = Input control voltage point 5
* F1   = Output frequency point 1
* F2   = Output frequency point 2
* F3   = Output frequency point 3
* F4   = Output frequency point 4
* F5   = Output frequency point 5
* Connections:
*               In+
*               | In-
*               | | Out+
*               | | | Out-
*               | | | |
.SUBCKT SINEVCO 1 2 3 4 PARAMS: C1=0 C2=1 C3=2 C4=3 C5=4
+ F1=0 F2=1k F3=2k F4=3k F5=4k LOW=-1 HIGH=1
A1 %vd(1,2) %vd(3,4) ASINEVCO
.MODEL ASINEVCO sine(cntl_array=[{C1} {C2} {C3} {C4} {C5}]
+ freq_array=[{F1} {F2} {F3} {F4} {F5}] out_low={LOW} out_high={HIGH})
.ENDS SINEVCO
```

If no overriding values for the parameters are entered on the **Parameters** tab of the *Sim Model* dialog the entries in the SPICE netlist would be:

```
*Schematic Netlist:
XV1 IN 0 OUT 0 SINEVCO
.
.
*Models and Subcircuit:
.SUBCKT SINEVCO 1 2 3 4 PARAMS: C1=0 C2=1 C3=2 C4=3 C5=4 F1=0 F2=1k F3=2k F4=3k
+ F5=4k LOW=-1 HIGH=1
A1 %vd(1,2) %vd(3,4) ASINEVCO
.MODEL ASINEVCO sine(cntl_array=[{C1} {C2} {C3} {C4} {C5}] freq_array=[{F1} {F2}
+ {F3} {F4} {F5}] out_low={LOW} out_high={HIGH})
.ENDS SINEVCO
```

The Netlister will evaluate the formulae in the sub-circuit definition using the default parameter values, as defined in the `SINEVCO.ckt` file.

Voltage Controlled Square Wave Oscillator



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL #|PARAMS:| ?LOW|LOW=@LOW| ?HIGH|HIGH=@HIGH|
?CYCLE|CYCLE=@CYCLE| ?RISE|RISE=@RISE| ?FALL|FALL=@FALL| ?C1|C1=@C1| ?F1|F1=@F1| ?C2|C2=@C2|
?F2|F2=@F2| ?C3|C3=@C3| ?F3|F3=@F3| ?C4|C4=@C4| ?F4|F4=@F4| ?C5|C5=@C5| ?F5|F5=@F5|
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Low - peak output low value (in Volts).

High - peak output high value (in Volts).

Cycle - duty cycle. Possible values can lie in the range 0 to 1.

Rise - rise time (in seconds).

Fall - fall time (in seconds).

C1 - input control voltage point 1 (in Volts).

F1 - output frequency point 1 (in Hertz).

C2 - input control voltage point 2 (in Volts).

F2 - output frequency point 2 (in Hertz).

C3 - input control voltage point 3 (in Volts).

F3 - output frequency point 3 (in Hertz).

C4 - input control voltage point 4 (in Volts).

F4 - output frequency point 4 (in Hertz).

C5 - input control voltage point 5 (in Volts).

F5 - output frequency point 5 (in Hertz).

Notes

The parameters `C1`, `C2`,`...` and `F1`, `F2`,`...` define the voltage to frequency conversion function. The C values define input voltage levels and the F values set the respective output frequencies generated for these input levels. Linear interpolation is used to define input/output values between the set points.

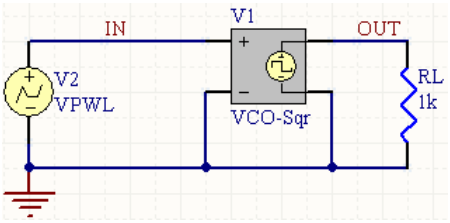
The voltage controlled square wave oscillator is not one of the built-in SPICE engine models. It is a complex device and, as such, is defined using the hierarchical sub-circuit syntax.

All of the parameters will normally have a default value assigned. The default should be applicable to most simulations. Generally you do not need to change this value. Entering a value for a parameter on the **Parameters** tab of the *Sim Model* dialog will override its specified value in the sub-circuit file.

To check the default values of the model, open the associated sub-circuit (`.ckt`) file. You can view the content of this file for the model specified on the **Model Kind** tab of the *Sim Model* dialog, by clicking on the **Model File** tab at the bottom of the dialog. The default parameter values are listed in the `.SUBCKT` line.

The simulation-ready voltage controlled square wave oscillator component (`VCO-Sqr`) can be found in the Simulation Special Function integrated library (`\Library\Simulation\Simulation Special Function.IntLib`).

Examples



Consider the voltage controlled square wave oscillator in the above image, with the following characteristics:

- Pin1 (positive controlling node) is connected to net `IN`
- Pin2 (negative controlling node) is connected to net `GND`
- Pin3 (positive output node) is connected to net `OUT`
- Pin4 (negative output node) is connected to net `GND`
- Designator is `V1`
- The linked simulation sub-circuit file is `SQRVCO.ckt`, with the following content:

```
*Voltage Controlled Square Wave Oscillator
*LOW   = Peak output low value
*HIGH  = Peak output high value
*CYCLE = Duty cycle
*RISE  = Rise time
*FALL  = Fall time
*C1    = Input control voltage point 1
*C2    = Input control voltage point 2
*C3    = Input control voltage point 3
*C4    = Input control voltage point 4
*C5    = Input control voltage point 5
*F1    = Output frequency point 1
*F2    = Output frequency point 2
*F3    = Output frequency point 3
*F4    = Output frequency point 4
*F5    = Output frequency point 5
* Connections:
*              In+
*              | In-
*              | | Out+
*              | | | Out-
*              | | | |
.SUBCKT SQRVCO 1 2 3 4 PARAMS: C1=0 C2=1 C3=2 C4=3 C5=4 F1=0 F2=1k
+ F3=2k F4=3k F5=4k LOW=0 HIGH=5 CYCLE=0.5 RISE=1u FALL=1u
A1 %vd(1,2) %vd(3,4) ASQRVCO
```

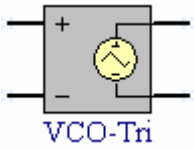
```
.MODEL ASQRVCO square(cntl_array=[{C1} {C2} {C3} {C4} {C5}]
+ freq_array=[{F1} {F2} {F3} {F4} {F5}] out_low={LOW}
+ out_high={HIGH} duty_cycle={CYCLE} rise_time={RISE} fall_time={FALL})
.ENDS SQRVCO
```

If no overriding values for the parameters are entered on the **Parameters** tab of the *Sim Model* dialog the entries in the SPICE netlist would be:

```
*Schematic Netlist:
XV1 IN 0 OUT 0 SQRVCO
.
.
*Models and Subcircuit:
.SUBCKT SQRVCO 1 2 3 4 PARAMS: C1=0 C2=1 C3=2 C4=3 C5=4 F1=0 F2=1k F3=2k F4=3k
+ F5=4k LOW=0 HIGH=5 CYCLE=0.5 RISE=1u FALL=1u
A1 %vd(1,2) %vd(3,4) ASQRVCO
.MODEL ASQRVCO square(cntl_array=[{C1} {C2} {C3} {C4} {C5}] freq_array=[{F1} {F2}
+ {F3} {F4} {F5}] out_low={LOW} out_high={HIGH} duty_cycle={CYCLE}
+ rise_time={RISE} fall_time={FALL})
.ENDS SQRVCO
```

The Netlister will evaluate the formulae in the sub-circuit definition using the default parameter values, as defined in the `SQRVCO.ckt` file.

Voltage Controlled Triangle Wave Oscillator



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL #|PARAMS:| ?LOW|LOW=@LOW| ?HIGH|HIGH=@HIGH| ?C1|C1=@C1|
?F1|F1=@F1| ?C2|C2=@C2| ?F2|F2=@F2| ?C3|C3=@C3| ?F3|F3=@F3| ?C4|C4=@C4| ?F4|F4=@F4|
?C5|C5=@C5| ?F5|F5=@F5| ?CYCLE|CYCLE=@CYCLE|
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Low - peak output low value (in Volts).

High - peak output high value (in Volts).

C1 - input control voltage point 1 (in Volts).

F1 - output frequency point 1 (in Hertz).

C2	-	input control voltage point 2 (in Volts).
F2	-	output frequency point 2 (in Hertz).
C3	-	input control voltage point 3 (in Volts).
F3	-	output frequency point 3 (in Hertz).
C4	-	input control voltage point 4 (in Volts).
F4	-	output frequency point 4 (in Hertz).
C5	-	input control voltage point 5 (in Volts).
F5	-	output frequency point 5 (in Hertz).
Cycle	-	duty cycle. Possible values can lie in the range 0 to 1.

Notes

The parameters `C1`, `C2`,`...` and `F1`, `F2`,`...` define the voltage to frequency conversion function. The C values define input voltage levels and the F values set the respective output frequencies generated for these input levels. Linear interpolation is used to define input/output values between the set points.

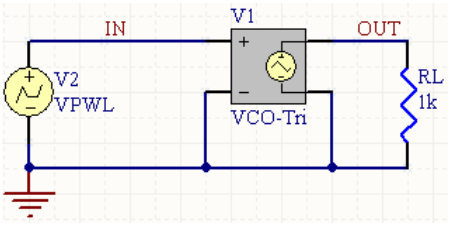
The voltage controlled triangle wave oscillator is not one of the built-in SPICE engine models. It is a complex device and, as such, is defined using the hierarchical sub-circuit syntax.

All of the parameters will normally have a default value assigned. The default should be applicable to most simulations. Generally you do not need to change this value. Entering a value for a parameter on the **Parameters** tab of the *Sim Model* dialog will override its specified value in the sub-circuit file.

To check the default values of the model, open the associated sub-circuit (`.ckt`) file. You can view the content of this file for the model specified on the **Model Kind** tab of the *Sim Model* dialog, by clicking on the **Model File** tab at the bottom of the dialog. The default parameter values are listed in the `.SUBCKT` line.

The simulation-ready voltage controlled triangle wave oscillator component (`VCO-Tri`) can be found in the Simulation Special Function integrated library (`\Library\Simulation\Simulation Special Function.IntLib`).

Examples



Consider the voltage controlled triangle wave oscillator in the above image, with the following characteristics:

- Pin1 (positive controlling node) is connected to net `IN`
- Pin2 (negative controlling node) is connected to net `GND`
- Pin3 (positive output node) is connected to net `OUT`
- Pin4 (negative output node) is connected to net `GND`
- Designator is `V1`
- The linked simulation sub-circuit file is `TRIVCO.ckt`, with the following content:

```
*Voltage Controlled Triangular Wave Oscillator
* LOW    = Peak output low value
* HIGH   = Peak output high value
* CYCLE = Duty cycle
* C1     = Input control voltage point 1
* C2     = Input control voltage point 2
* C3     = Input control voltage point 3
* C4     = Input control voltage point 4
```

```
* C5    = Input control voltage point 5
* F1    = Output frequency point 1
* F2    = Output frequency point 2
* F3    = Output frequency point 3
* F4    = Output frequency point 4
* F5    = Output frequency point 5
* Connections:
*               In+
*               | In-
*               | | Out+
*               | | | Out-
*               | | | |
.SUBCKT TRIVCO 1 2 3 4 PARAMS: C1=0 C2=1 C3=2 C4=3 C5=4
+ F1=0 F2=1k F3=2k F4=3k F5=4k LOW=-5 HIGH=5 CYCLE=0.5
A1 %vd(1,2) %vd(3,4) ATRIVCO
.MODEL ATRIVCO triangle(cntl_array=[{C1} {C2} {C3} {C4} {C5}]
+ freq_array=[{F1} {F2} {F3} {F4} {F5}] out_low={LOW}
+ out_high={HIGH} duty_cycle={CYCLE})
.ENDS TRIVCO
```

If no overriding values for the parameters are entered on the **Parameters** tab of the *Sim Model* dialog the entries in the SPICE netlist would be:

```
*Schematic Netlist:
XV1 IN 0 OUT 0 TRIVCO
.
.
*Models and Subcircuit:
.SUBCKT TRIVCO 1 2 3 4 PARAMS: C1=0 C2=1 C3=2 C4=3 C5=4 F1=0 F2=1k F3=2k F4=3k
+ F5=4k LOW=-5 HIGH=5 CYCLE=0.5
A1 %vd(1,2) %vd(3,4) ATRIVCO
.MODEL ATRIVCO triangle(cntl_array=[{C1} {C2} {C3} {C4} {C5}] freq_array=[{F1}
+ {F2} {F3} {F4} {F5}] out_low={LOW} out_high={HIGH} duty_cycle={CYCLE})
.ENDS TRIVCO
```

The Netlister will evaluate the formulae in the sub-circuit definition using the default parameter values, as defined in the `TRIVCO.ckt` file.

Math Functions

The simulation-ready components in this section provide for mathematical functions in the following categories:

Absolute Value	• *Current*	• *Voltage (Single-Ended)*
• *Voltage (Differential)*	**Addition**	• *Current*
• *Voltage (Single-Ended)*	• *Voltage (Differential)*	**Arc Cosine**
• *Current*	• *Voltage (Single-Ended)*	• *Voltage (Differential)*
Arc Sine	• *Current*	• *Voltage (Single-Ended)*
• *Voltage (Differential)*	**Arc Tangent**	• *Current*
• *Voltage (Single-Ended)*	• *Voltage (Differential)*	**Cosine**
• *Current*	• *Voltage (Single-Ended)*	• *Voltage (Differential)*
Division	• *Current*	• *Voltage (Single-Ended)*
• *Voltage (Differential)*	**Exponential**	• *Current*
• *Voltage (Single-Ended)*	• *Voltage (Differential)*	**Hyperbolic Arc Cosine**
• *Current*	• *Voltage (Single-Ended)*	• *Voltage (Differential)*
Hyperbolic Arc Sine	• *Current*	• *Voltage (Single-Ended)*
• *Voltage (Differential)*	**Hyperbolic Arc Tangent**	• *Current*
• *Voltage (Single-Ended)*	• *Voltage (Differential)*	**Hyperbolic Cosine**
• *Current*	• *Voltage (Single-Ended)*	• *Voltage (Differential)*
Hyperbolic Sine	• *Current*	• *Voltage (Single-Ended)*
• *Voltage (Differential)*	**Logarithm (Base 10)**	• *Current*
• *Voltage (Single-Ended)*	• *Voltage (Differential)*	**Multiplication**
• *Current*	• *Voltage (Single-Ended)*	• *Voltage (Differential)*
Natural Logarithm (Base e)	• *Current*	• *Voltage (Single-Ended)*
• *Voltage (Differential)*	**Sine**	• *Current*
• *Voltage (Single-Ended)*	• *Voltage (Differential)*	**Square Root**
• *Current*	• *Voltage (Single-Ended)*	• *Voltage (Differential)*
Subtraction	• *Current*	• *Voltage (Single-Ended)*
• *Voltage (Differential)*	**Tangent**	• *Current*
• *Voltage (Single-Ended)*	• *Voltage (Differential)*	**Unary Minus**
• *Current*	• *Voltage (Single-Ended)*	• *Voltage (Differential)*

Notes

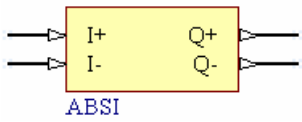
Functions in each category are available for operation with voltage (both differential and single-ended) and current.

The models for these devices are not built-in SPICE engine models. They are complex devices and, as such, are defined using the hierarchical sub-circuit syntax.

All mathematical function components can be found in the Simulation Math Function integrated library (`\Library\Simulation\Simulation Math Function.IntLib`).

Absolute Value

Absolute Value of Current



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ABSI

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

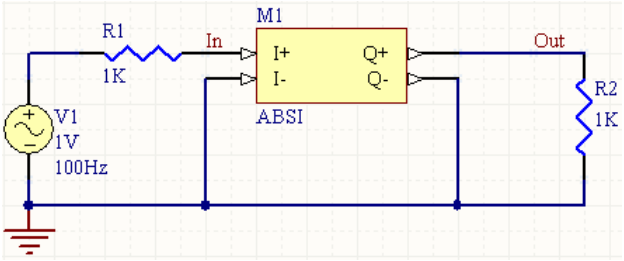
None

Notes

The content of the sub-circuit file (`ABSI.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Absolute value of Current
.SUBCKT ABSI 1 2 3 4
VX 1 2 0
BX 4 3 I=ABS(I(VX))
.ENDS ABSI
```

Examples

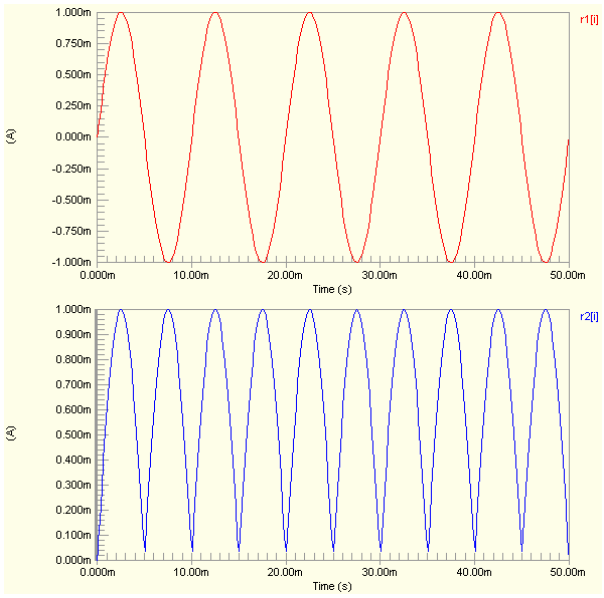


Consider the circuit in the image above. With respect to the ABSI component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN 0 OUT 0 ABSI
.
.
*Models and Subcircuit:
.SUBCKT ABSI 1 2 3 4
VX 1 2 0
```

```
BX 4 3 I=ABS(I(VX))
.ENDS ABSI
```

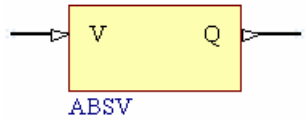
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Absolute Value of Voltage (Single-Ended Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ABSV

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @MODEL
```

Parameters (definable at component level)

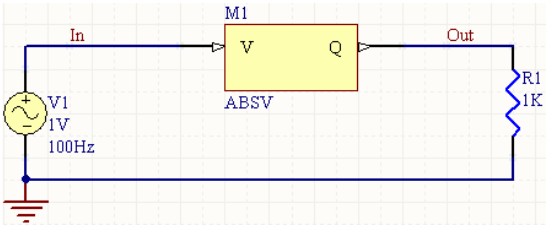
None

Notes

The content of the sub-circuit file (`ABSV.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Absolute value of Voltage
.SUBCKT ABSV 1 2
BX 2 0 V=ABS(V(1))
.ENDS ABSV
```

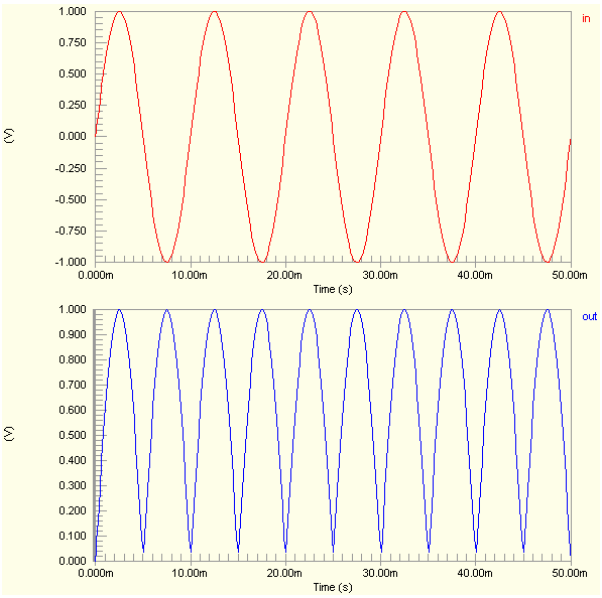
Examples



Consider the circuit in the image above. With respect to the ABSV component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN OUT ABSV
.
.
*Models and Subcircuit:
.SUBCKT ABSV 1 2
BX 2 0 V=ABS(V(1))
.ENDS ABSV
```

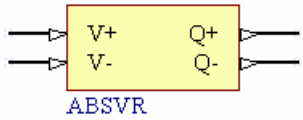
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Absolute Value of Voltage (Differential Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ABSVR

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

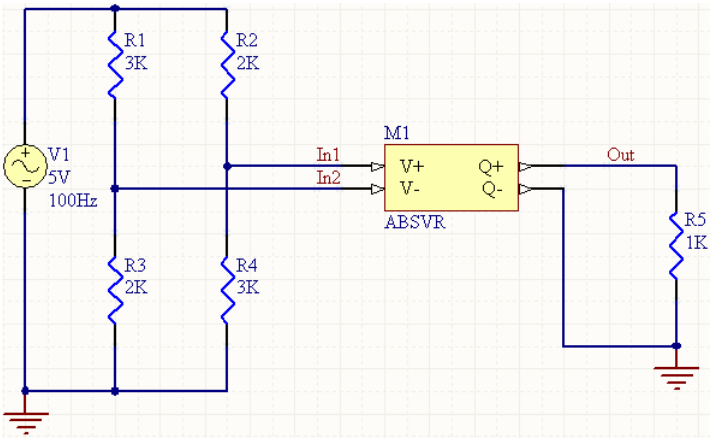
None

Notes

The content of the sub-circuit file (`ABSVR.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Absolute value of Voltage -- pkg:ABS(V)
.SUBCKT ABSVR 1 2 3 4
BX 3 4 V=ABS(V(1,2))
.ENDS ABSVR
```

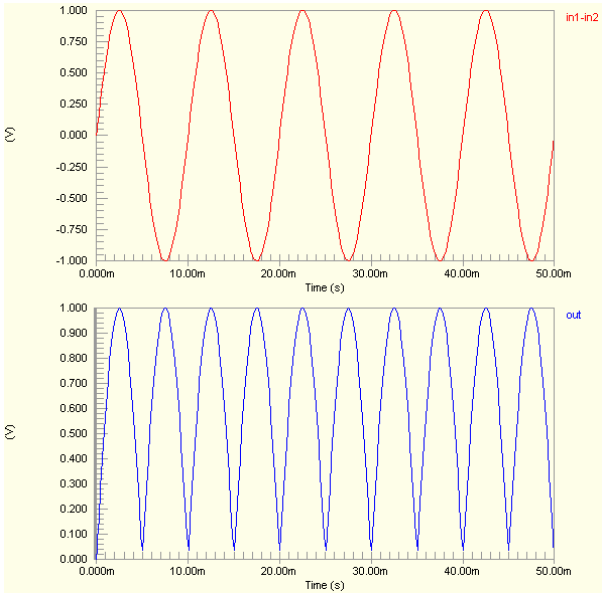
Examples



Consider the circuit in the image above. With respect to the ABSVR component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN1 IN2 OUT 0 ABSVR
.
.
*Models and Subcircuit:
.SUBCKT ABSVR 1 2 3 4
BX 3 4 V=ABS(V(1,2))
.ENDS ABSVR
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

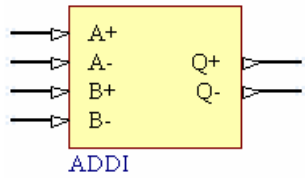


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Addition

Addition of Currents



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ADDI

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 %5 %6 @MODEL
```

Parameters (definable at component level)

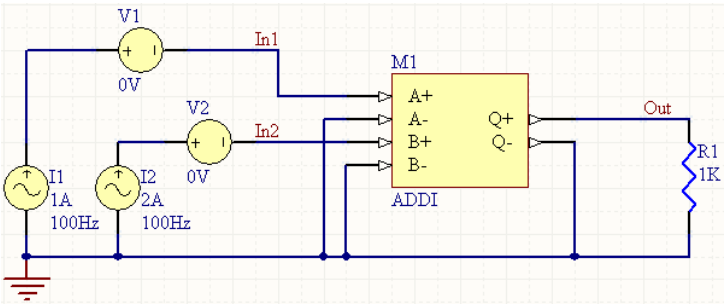
None

Notes

The content of the sub-circuit file (`ADDI.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Add Currents
.SUBCKT ADDI 1 2 3 4 5 6
VA 1 2 0
VB 3 4 0
BX 6 5 I=I(VA)+I(VB)
.ENDS ADDI
```

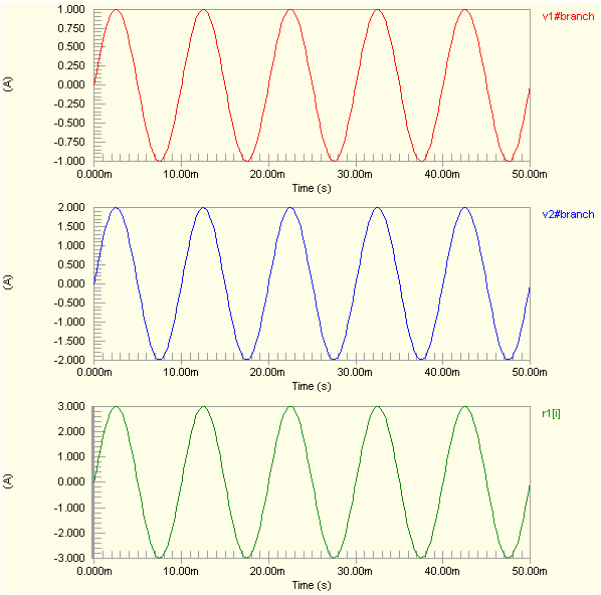
Examples



Consider the circuit in the image above. With respect to the ADDI component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN1 0 IN2 0 OUT 0 ADDI
.
.
*Models and Subcircuit:
.SUBCKT ADDI 1 2 3 4 5 6
VA 1 2 0
VB 3 4 0
BX 6 5 I=I(VA)+I(VB)
.ENDS ADDI
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

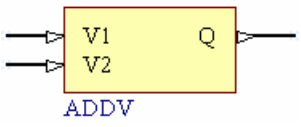


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`

- **Transient Max Step Time** – set to `200.0u`.

Addition of Voltages (Single-Ended Inputs)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ADDV

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 @MODEL
```

Parameters (definable at component level)

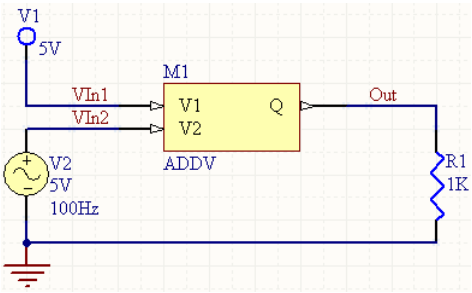
None

Notes

The content of the sub-circuit file (`ADDV.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Add Voltages
.SUBCKT ADDV 1 2 3
BX 3 0 V=V(1)+V(2)
.ENDS ADDV
```

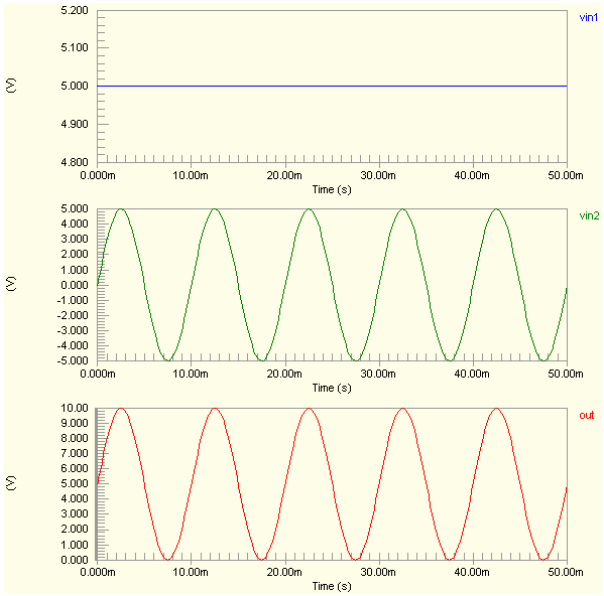
Examples



Consider the circuit in the previous image. With respect to the ADDV component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 VIN1 VIN2 OUT ADDV
.
.
*Models and Subcircuit:
.SUBCKT ADDV 1 2 3
BX 3 0 V=V(1)+V(2)
.ENDS ADDV
```

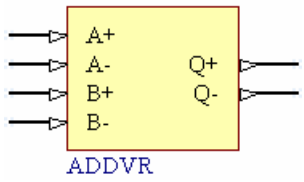
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Addition of Voltages (Differential Inputs)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ADDVR

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 %5 %6 @MODEL
```

Parameters (definable at component level)

None

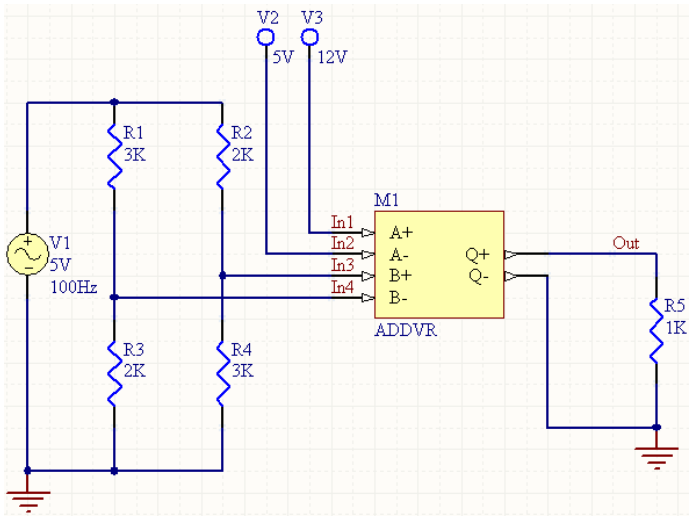
Notes

The content of the sub-circuit file (`ADDVR.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Add Voltages
.SUBCKT ADDVR 1 2 3 4 5 6
```

```
BX 5 6 V=V(1,2)+V(3,4)
.ENDS ADDVR
```

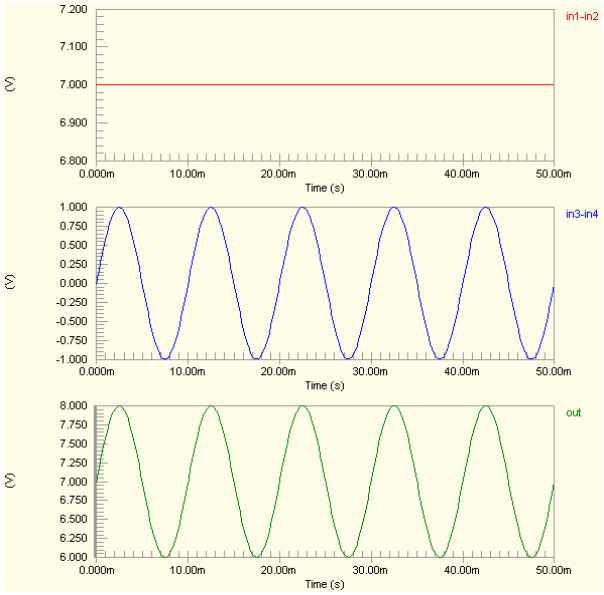
Examples



Consider the circuit in the image above. With respect to the ADDVR component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN1 IN2 IN3 IN4 OUT 0 ADDVR
.
.
*Models and Subcircuit:
.SUBCKT ADDVR 1 2 3 4 5 6
BX 5 6 V=V(1,2)+V(3,4)
.ENDS ADDVR
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



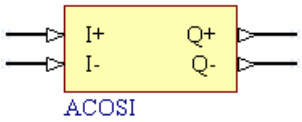
In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`

- **Transient Max Step Time** – set to `200.0u`.

Arc Cosine

Arc Cosine of Current



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ACOSI

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

None

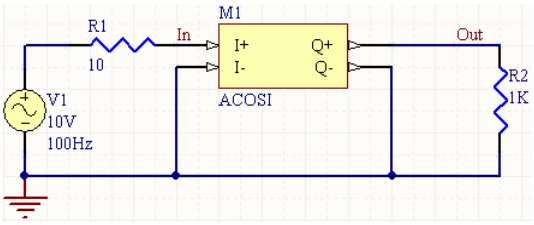
Notes

The content of the sub-circuit file (`ACOSI.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Arc cosine of Current
.SUBCKT ACOSI 1 2 3 4
VX 1 2 0
BX 4 3 I=ACOS(I(VX))
.ENDS ACOSI
```

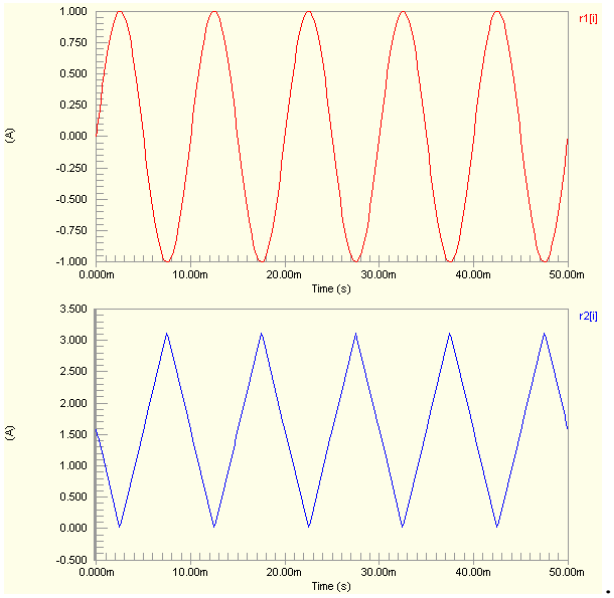
The resulting current is the value expressed in radians.

Examples



Consider the circuit in the image above. With respect to the ACOSI component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN 0 OUT 0 ACOSI
.
```



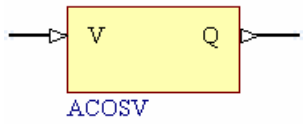

```
*Models and Subcircuit:
.SUBCKT ACOSI 1 2 3 4
VX 1 2 0
BX 4 3 I=ACOS(I(VX))
.ENDS ACOSI
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Arc Cosine of Voltage (Single-Ended Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ACOSV

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @MODEL
```

Parameters (definable at component level)

None

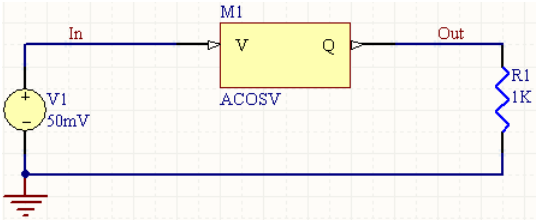
Notes

The content of the sub-circuit file (`ACOSV.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Arc cosine of Voltage
.SUBCKT ACOSV 1 2
BX 2 0 V=ACOS(V(1))
.ENDS ACOSV
```

The resulting voltage is the value expressed in radians.

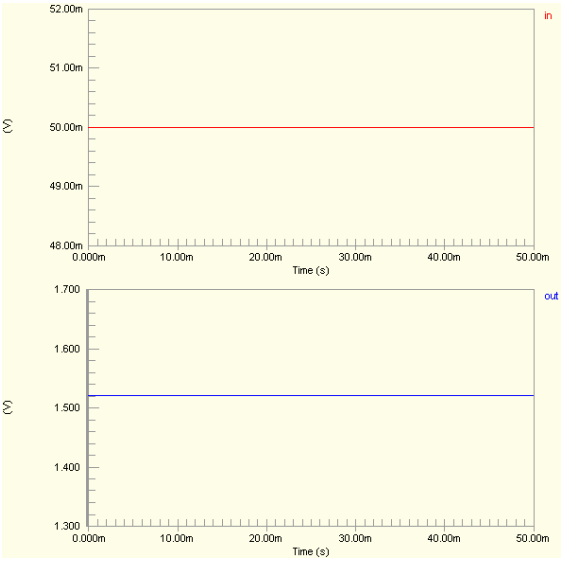
Examples



Consider the circuit in the image above. With respect to the ACOSV component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN OUT ACOSV
.
.
*Models and Subcircuit:
.SUBCKT ACOSV 1 2
BX 2 0 V=ACOS(V(1))
.ENDS ACOSV
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

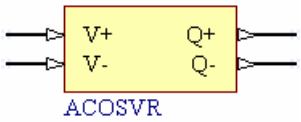


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`

- **Transient Max Step Time** – set to `200.0u`.

Arc Cosine of Voltage (Differential Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ACOSVR

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

None

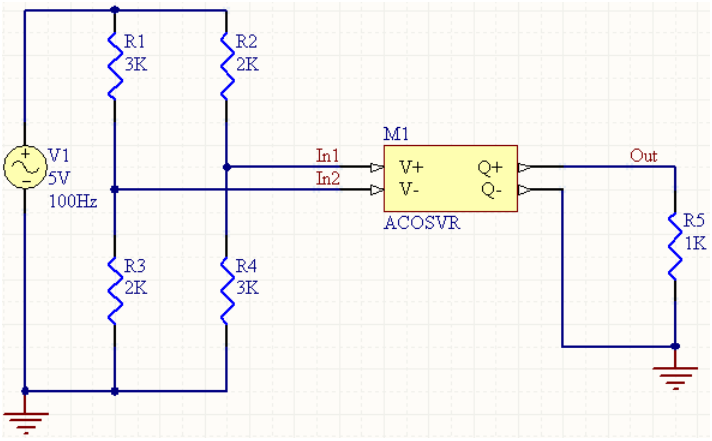
Notes

The content of the sub-circuit file (`ACOSVR.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Arc cosine of Voltage
.SUBCKT ACOSVR 1 2 3 4
BX 3 4 V=ACOS(V(1,2))
.ENDS ACOSVR
```

The resulting voltage is the value expressed in radians.

Examples

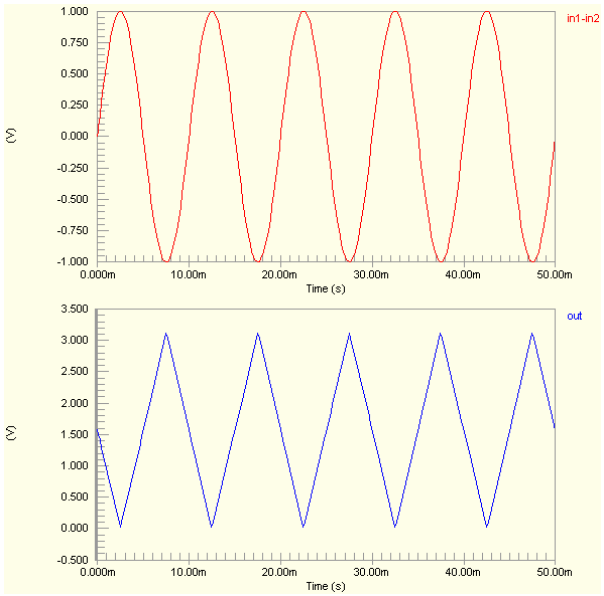


Consider the circuit in the image above. With respect to the ACOSVR component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN1 IN2 OUT 0 ACOSVR
.
.
*Models and Subcircuit:
```

```
.SUBCKT ACOSVR 1 2 3 4
BX 3 4 V=ACOS(V(1,2))
.ENDS ACOSVR
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

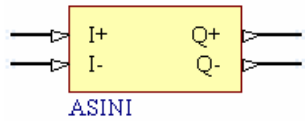


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Arc Sine

Arc Sine of Current



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ASINI

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

None

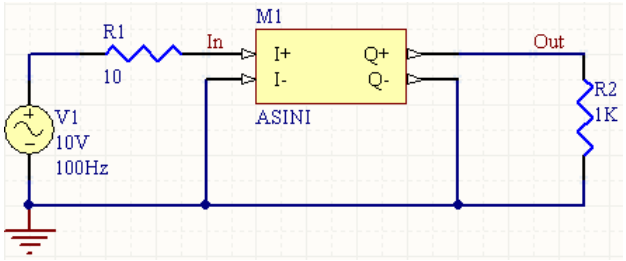
Notes

The content of the sub-circuit file (`ASINI.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Arc sine of Current
.SUBCKT ASINI 1 2 3 4
VX 1 2 0
BX 4 3 I=ASIN(I(VX))
.ENDS ASINI
```

The resulting current is the value expressed in radians.

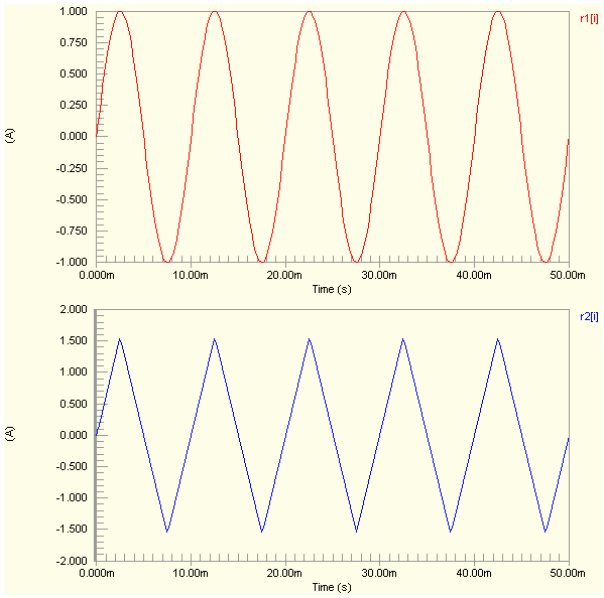
Examples



Consider the circuit in the image above. With respect to the ASINI component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN 0 OUT 0 ASINI
.
.
*Models and Subcircuit:
.SUBCKT ASINI 1 2 3 4
VX 1 2 0
BX 4 3 I=ASIN(I(VX))
.ENDS ASINI
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

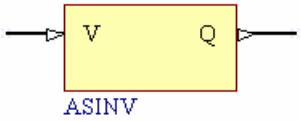


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`

- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Arc Sine of Voltage (Single-Ended Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ASINV

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @MODEL
```

Parameters (definable at component level)

None

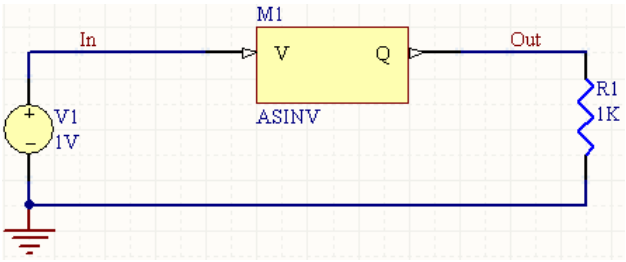
Notes

The content of the sub-circuit file (`ASINV.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Arc sine of Voltage
.SUBCKT ASINV 1 2
BX 2 0 V=ASIN(V(1))
.ENDS ASINV
```

The resulting voltage is the value expressed in radians.

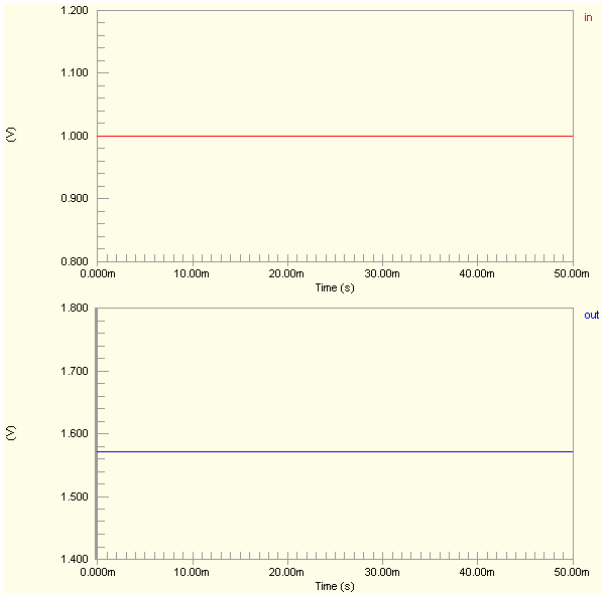
Examples



Consider the circuit in the image above. With respect to the ASINV component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN OUT ASINV
.
.
*Models and Subcircuit:
.SUBCKT ASINV 1 2
BX 2 0 V=ASIN(V(1))
.ENDS ASINV
```

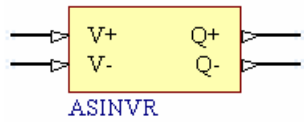
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Arc Sine of Voltage (Differential Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ASINVR

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

None

Notes

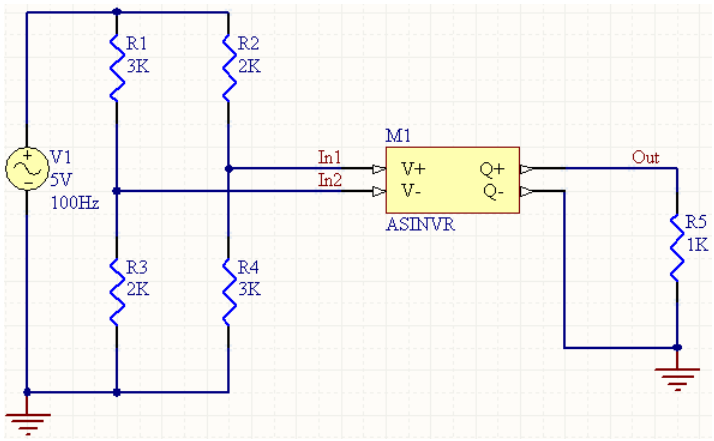
The content of the sub-circuit file (`ASINVR.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Arc sine of Voltage
.SUBCKT ASINVR 1 2 3 4
```

```
BX 3 4 V=ASIN(V(1,2))
.ENDS ASINVR
```

The resulting voltage is the value expressed in radians.

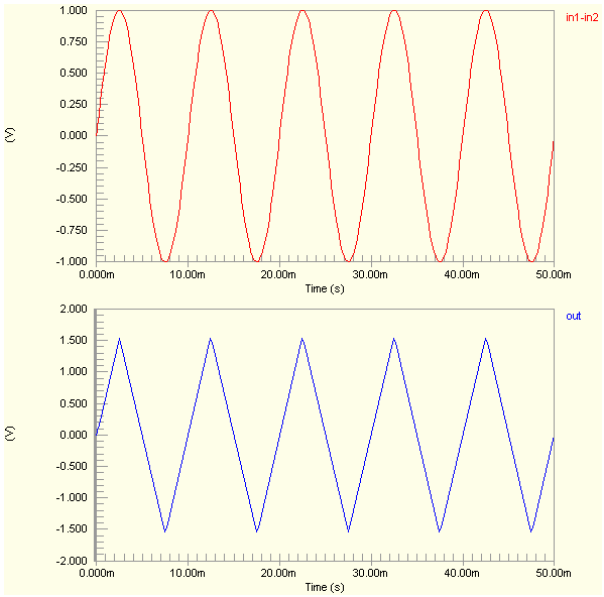
Examples



Consider the circuit in the image above. With respect to the ASINVR component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN1 IN2 OUT 0 ASINVR
.
.
*Models and Subcircuit:
.SUBCKT ASINVR 1 2 3 4
BX 3 4 V=ASIN(V(1,2))
.ENDS ASINVR
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

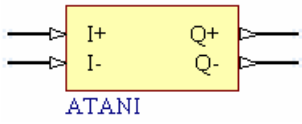


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Arc Tangent

Arc Tangent of Current



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ATANI

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

None

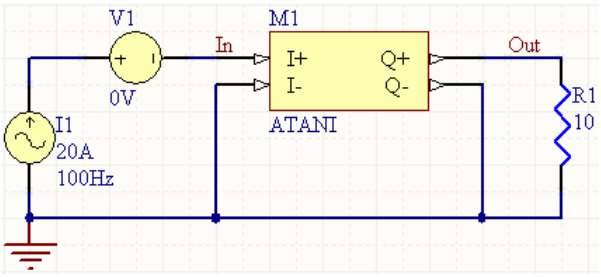
Notes

The content of the sub-circuit file (`ATANI.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Arc tangent of Current
.SUBCKT ATANI 1 2 3 4
VX 1 2 0
BX 4 3 I=ATAN(I(VX))
.ENDS ATANI
```

The resulting current is the value expressed in radians.

Examples

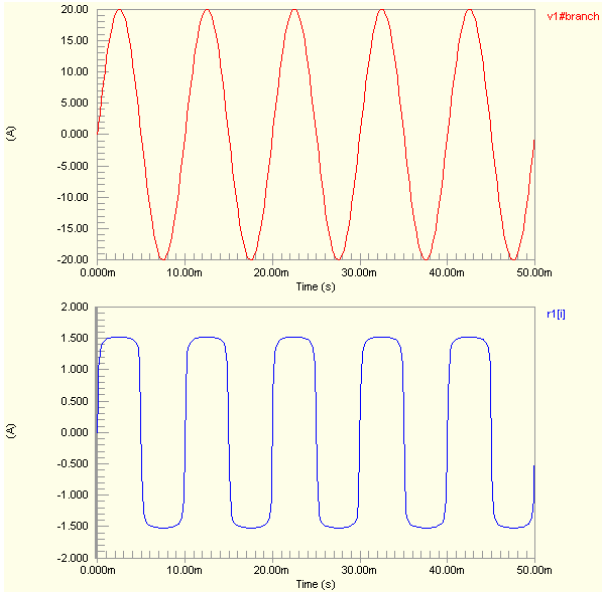


Consider the circuit in the image above. With respect to the ATANI component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN 0 OUT 0 ATANI
.
.
*Models and Subcircuit:
.SUBCKT ATANI 1 2 3 4
VX 1 2 0
BX 4 3 I=ATAN(I(VX))
```

```
.ENDS ATANI
```

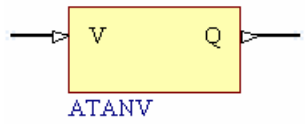
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Arc Tangent of Voltage (Single-Ended Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ATANV

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @MODEL
```

Parameters (definable at component level)

None

Notes

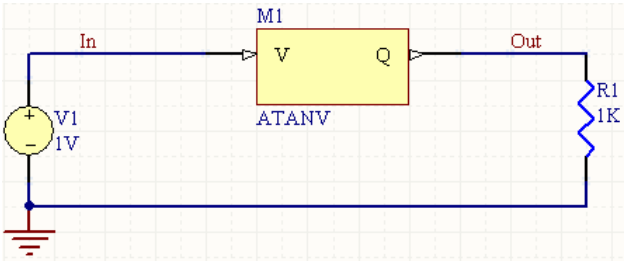
The content of the sub-circuit file (`ATANV.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Arc tangent of Voltage
```

```
.SUBCKT ATANV 1 2
BX 2 0 V=ATAN(V(1))
.ENDS ATANV
```

The resulting voltage is the value expressed in radians.

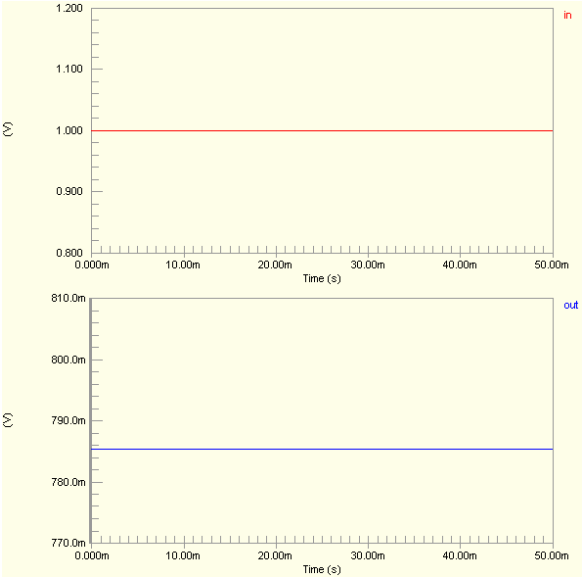
Examples



Consider the circuit in the image above. With respect to the ATANV component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN OUT ATANV
.
.
*Models and Subcircuit:
.SUBCKT ATANV 1 2
BX 2 0 V=ATAN(V(1))
.ENDS ATANV
```

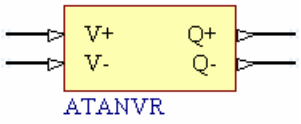
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Arc Tangent of Voltage (Differential Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ATANVR

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

None

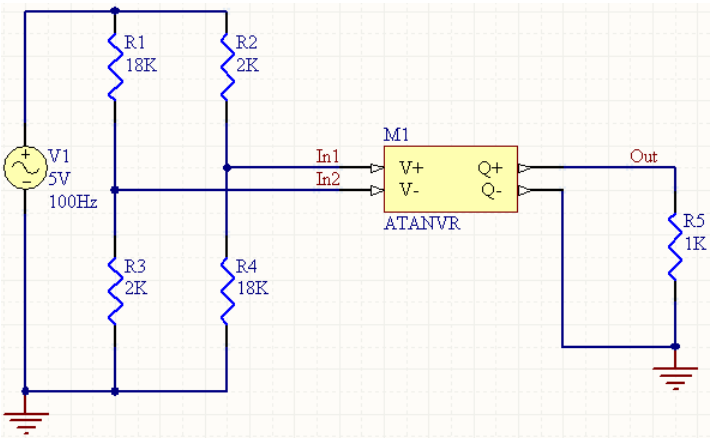
Notes

The content of the sub-circuit file (`ATANVR.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Arc tangent of Voltage -- pkg:ATAN(V)
.SUBCKT ATANVR 1 2 3 4
BX 3 4 V=ATAN(V(1,2))
.ENDS ATANVR
```

The resulting voltage is the value expressed in radians.

Examples

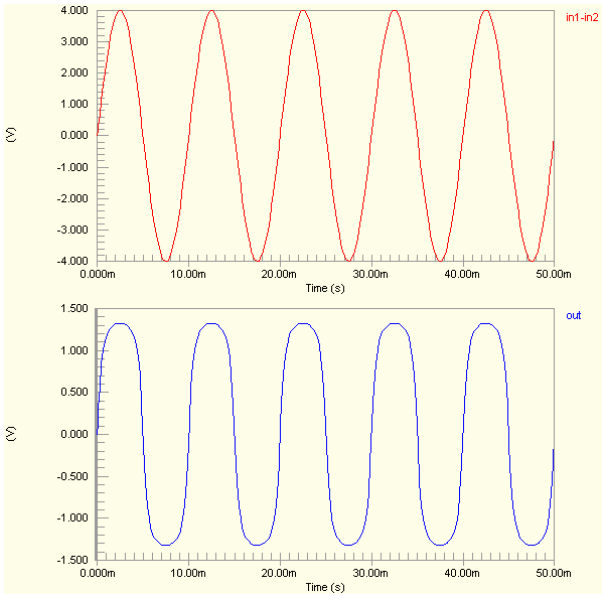


Consider the circuit in the image above. With respect to the ATANVR component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN1 IN2 OUT 0 ATANVR
.
.
*Models and Subcircuit:
.SUBCKT ATANVR 1 2 3 4
```

```
BX 3 4 V=ATAN(V(1,2))
.ENDS ATANVR
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

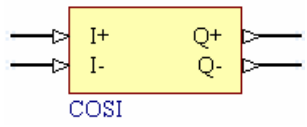


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Cosine

Cosine of Current



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

COSI

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

None

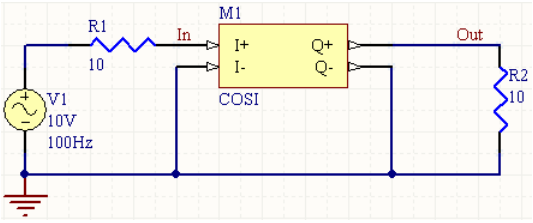
Notes

The content of the sub-circuit file (`COSI.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Cosine of Current
.SUBCKT COSI 1 2 3 4
VX 1 2 0
BX 4 3 I=COS(I(VX))
.ENDS COSI
```

The resulting current is the value expressed in radians.

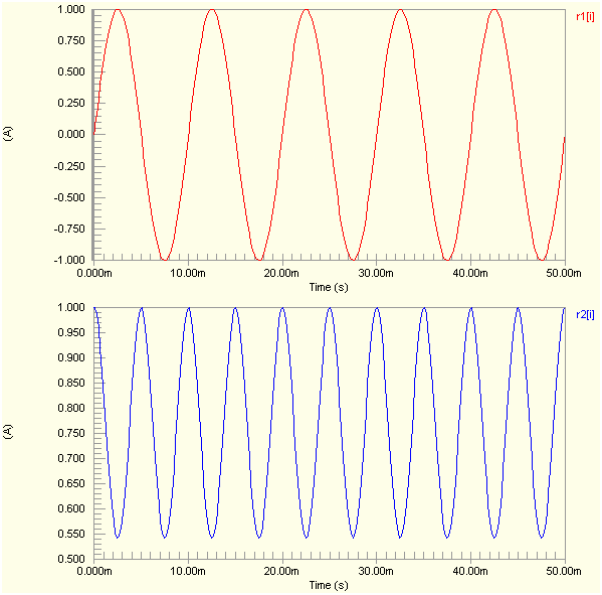
Examples



Consider the circuit in the image above. With respect to the COSI component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN 0 OUT 0 COSI
.
.
*Models and Subcircuit:
.SUBCKT COSI 1 2 3 4
VX 1 2 0
BX 4 3 I=COS(I(VX))
.ENDS COSI
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

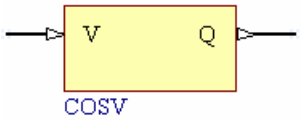


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`

- **Transient Max Step Time** – set to `200.0u`.

Cosine of Voltage (Single-Ended Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

COSV

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @MODEL
```

Parameters (definable at component level)

None

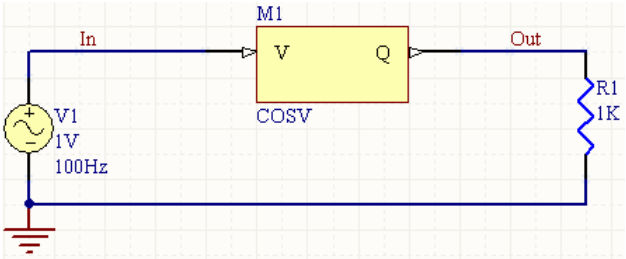
Notes

The content of the sub-circuit file (`COSV.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Cosine of Voltage
.SUBCKT COSV 1 2
BX 2 0 V=COS(V(1))
.ENDS COSV
```

The resulting voltage is the value expressed in radians.

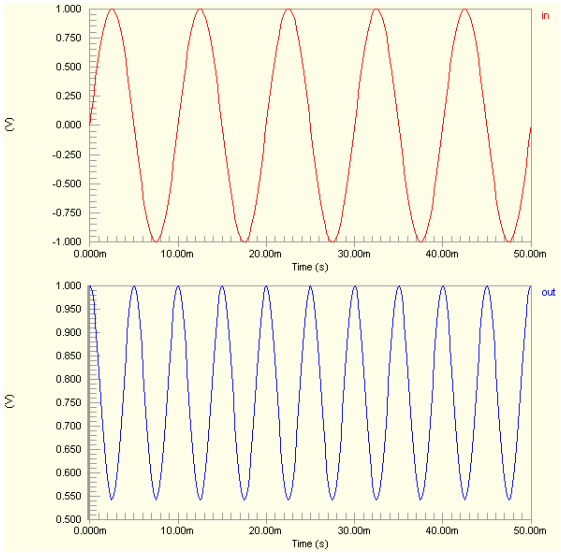
Examples



Consider the circuit in the image above. With respect to the COSV component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN OUT COSV
.
.
*Models and Subcircuit:
.SUBCKT COSV 1 2
BX 2 0 V=COS(V(1))
.ENDS COSV
```

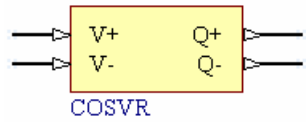
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Cosine of Voltage (Differential Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

COSVR

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

None

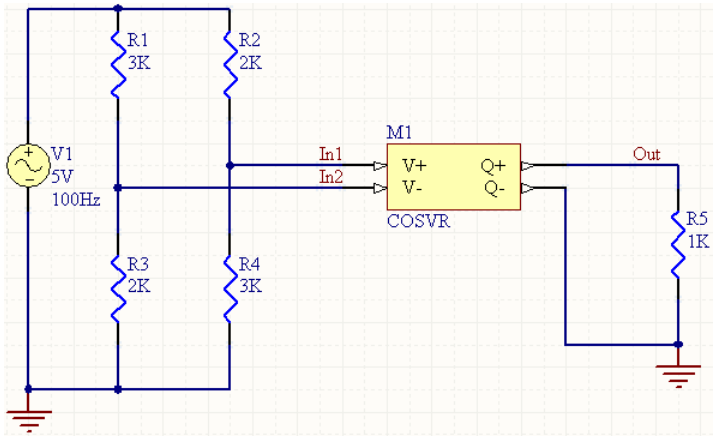
Notes

The content of the sub-circuit file (`COSVR.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Cosine of Voltage
.SUBCKT COSVR 1 2 3 4
BX 3 4 V=COS(V(1,2))
.ENDS COSVR
```

The resulting voltage is the value expressed in radians.

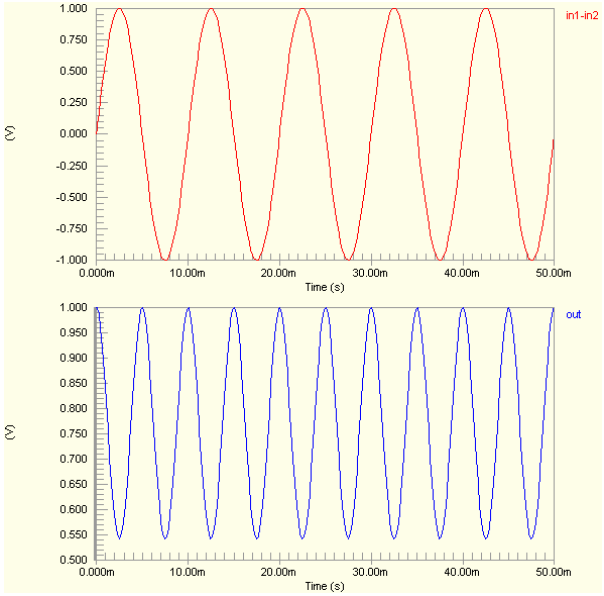
Examples



Consider the circuit in the image above. With respect to the COSVR component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN1 IN2 OUT 0 COSVR
.
.
*Models and Subcircuit:
.SUBCKT COSVR 1 2 3 4
BX 3 4 V=COS(V(1,2))
.ENDS COSVR
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

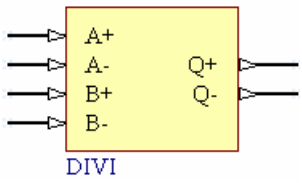


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Division

Division of Currents



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

DIVI

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 %5 %6 @MODEL
```

Parameters (definable at component level)

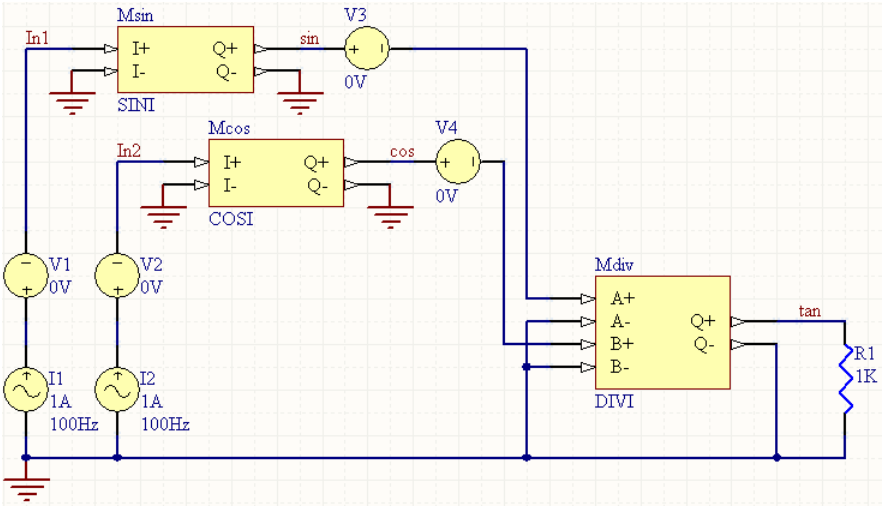
None

Notes

The content of the sub-circuit file (`DIVI.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Divide Currents
.SUBCKT DIVI 1 2 3 4 5 6
VA 1 2 0
VB 3 4 0
BX 6 5 I=I(VA)/I(VB)
.ENDS DIVI
```

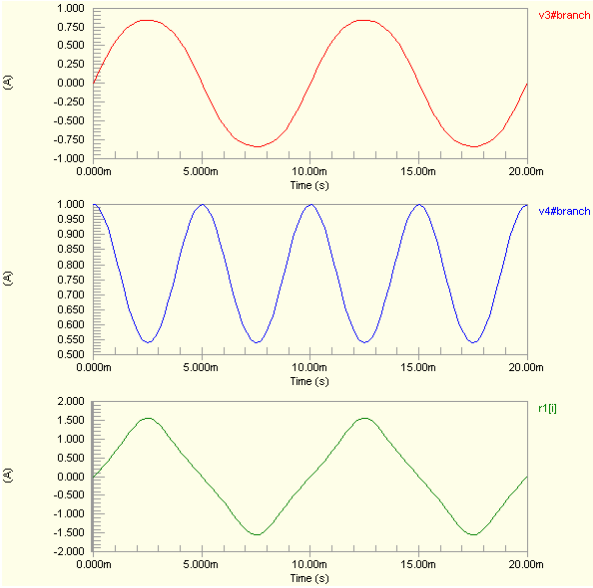
Examples



Consider the circuit in the image above. With respect to the DIVI component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XMdiv NetMdiv_1 0 NetMdiv_3 0 TAN 0 DIVI
.
.
*Models and Subcircuit:
.SUBCKT DIVI 1 2 3 4 5 6
VA 1 2 0
VB 3 4 0
BX 6 5 I=I(VA)/I(VB)
.ENDS DIVI
```

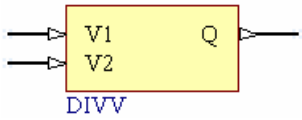
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Division of Voltages (Single-Ended Inputs)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

DIVV

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 @MODEL
```

Parameters (definable at component level)

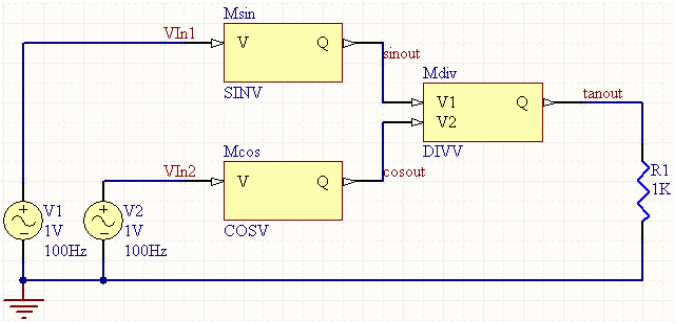
None

Notes

The content of the sub-circuit file (`DIVV.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Divide Voltages
.SUBCKT DIVV 1 2 3
BX 3 0 V=V(1)/V(2)
.ENDS DIVV
```

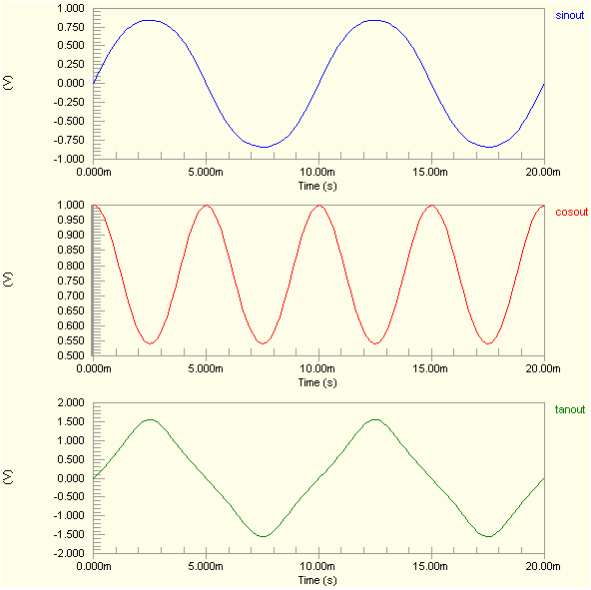
Examples



Consider the circuit in the previous mage. With respect to the DIVV component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XMdiv SINOUT COSOUT TANOUT DIVV
.
.
*Models and Subcircuit:
.SUBCKT DIVV 1 2 3
BX 3 0 V=V(1)/V(2)
.ENDS DIVV
```

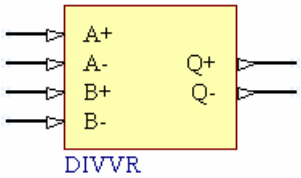
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Division of Voltages (Differential Inputs)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

DIVVR

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 %5 %6 @MODEL
```

Parameters (definable at component level)

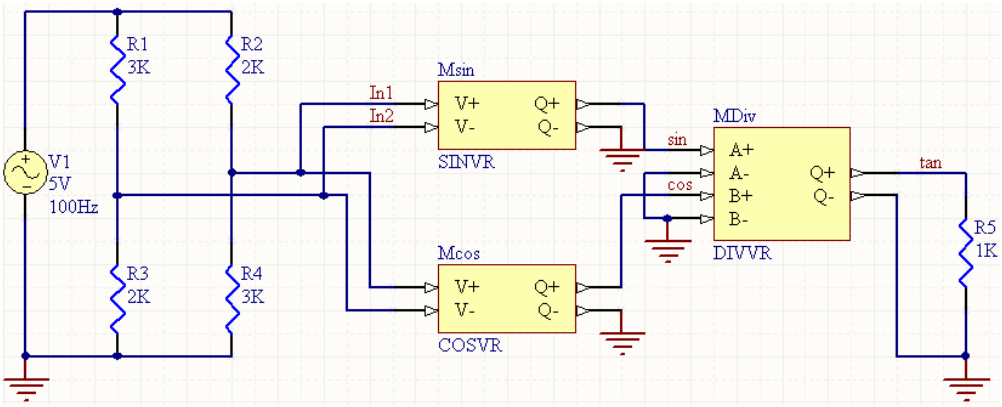
None

Notes

The content of the sub-circuit file (`DIVVR.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Divide Voltages
.SUBCKT DIVVR 1 2 3 4 5 6
BX 5 6 V=V(1,2)/V(3,4)
.ENDS DIVVR
```

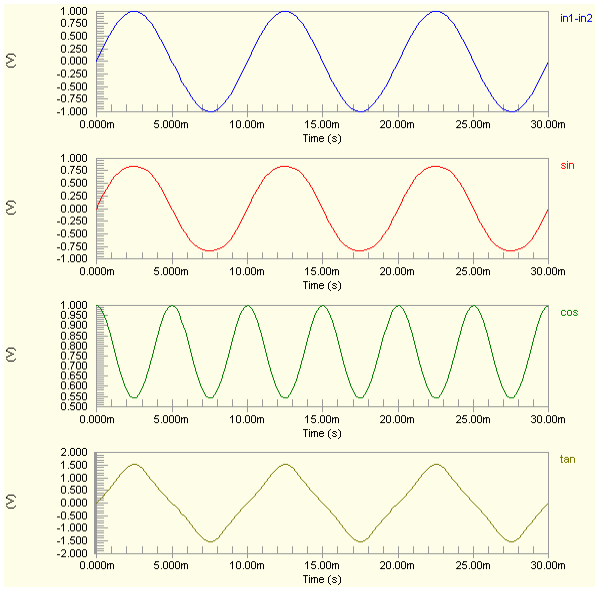
Examples



Consider the circuit in the image above. With respect to the DIVVR component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XMDiv SIN 0 COS 0 TAN 0 DIVVR
.
.
*Models and Subcircuit:
.SUBCKT DIVVR 1 2 3 4 5 6
BX 5 6 V=V(1,2)/V(3,4)
.ENDS DIVVR
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

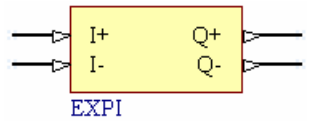


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Exponential

Exponential of Current



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

EXPI

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

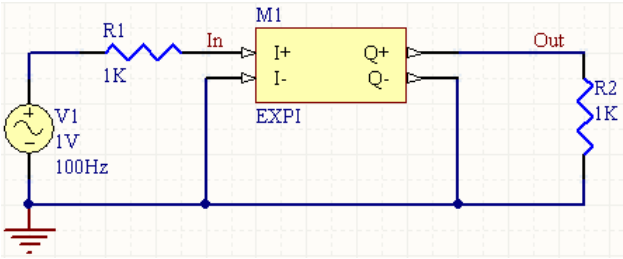
None

Notes

The content of the sub-circuit file (`EXPI.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Exponential of Current
.SUBCKT EXPI 1 2 3 4
VX 1 2 0
BX 4 3 I=EXP(I(VX))
.ENDS EXPI
```

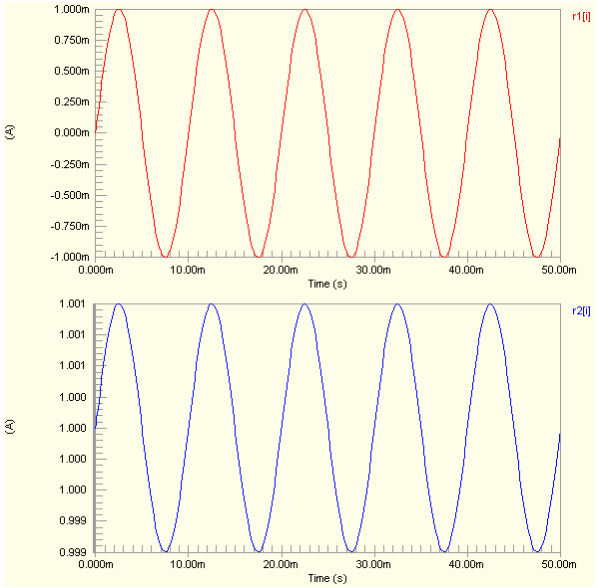
Examples



Consider the circuit in the image above. With respect to the EXPI component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN 0 OUT 0 EXPI
.
.
*Models and Subcircuit:
.SUBCKT EXPI 1 2 3 4
VX 1 2 0
BX 4 3 I=EXP(I(VX))
.ENDS EXPI
```

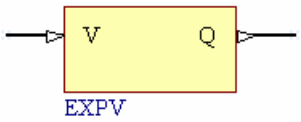
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Exponential of Voltage (Single-Ended Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

EXPV

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @MODEL
```

Parameters (definable at component level)

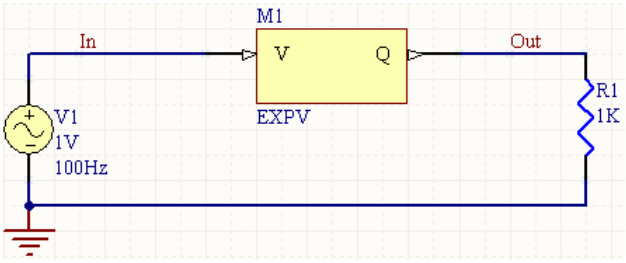
None

Notes

The content of the sub-circuit file (`EXPV.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Exponential of Voltage
.SUBCKT EXPV 1 2
BX 2 0 V=EXP(V(1))
.ENDS EXPV
```

Examples

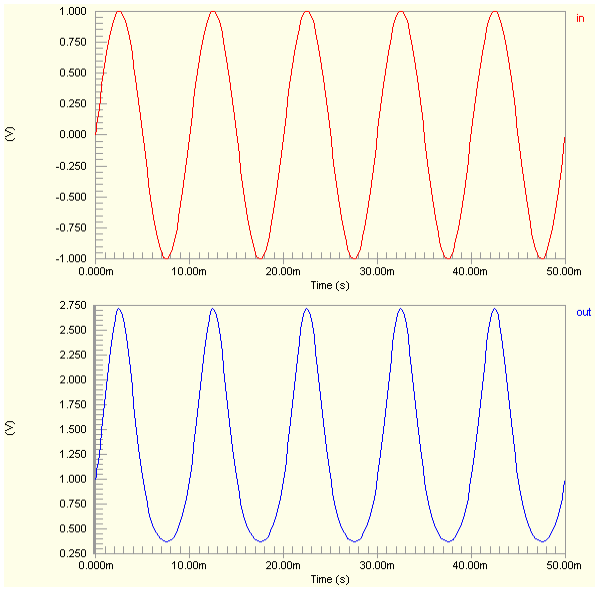


Consider the circuit in the image above. With respect to the EXPV component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN OUT EXPV
.
.
*Models and Subcircuit:
```

```
.SUBCKT EXPV 1 2
BX 2 0 V=EXP(V(1))
.ENDS EXPV
```

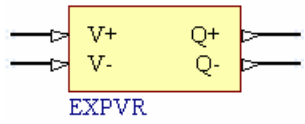
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Exponential of Voltage (Differential Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

EXPVR

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

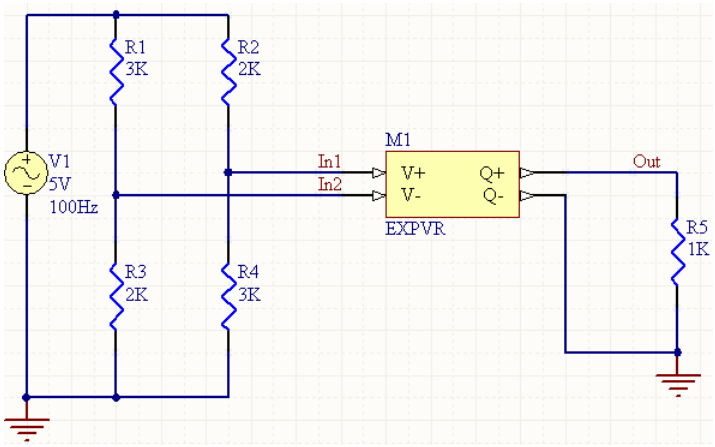
None

Notes

The content of the sub-circuit file (`EXPVR.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Exponential of Voltage
.SUBCKT EXPVR 1 2 3 4
BX 3 4 V=EXP(V(1,2))
.ENDS EXPVR
```

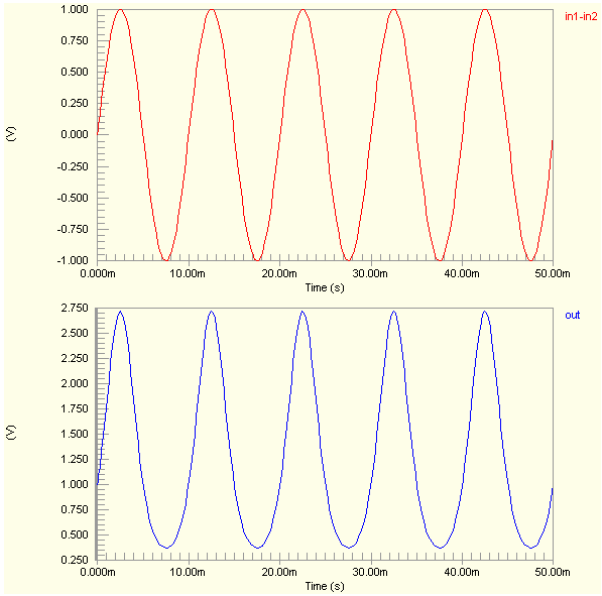
Examples



Consider the circuit in the image above. With respect to the EXPVR component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN1 IN2 OUT 0 EXPVR
.
.
*Models and Subcircuit:
.SUBCKT EXPVR 1 2 3 4
BX 3 4 V=EXP(V(1,2))
.ENDS EXPVR
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

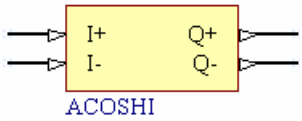


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Hyperbolic Arc Cosine

Hyperbolic Arc Cosine of Current



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ACOSHI

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

None

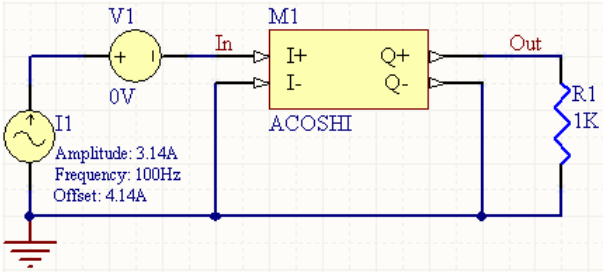
Notes

The content of the sub-circuit file (`ACOSHI.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Hyperbolic arc cosine of Current
.SUBCKT ACOSHI 1 2 3 4
VX 1 2 0
BX 4 3 I=ACOSH(I(VX))
.ENDS ACOSHI
```

The resulting current is the value expressed in radians.

Examples

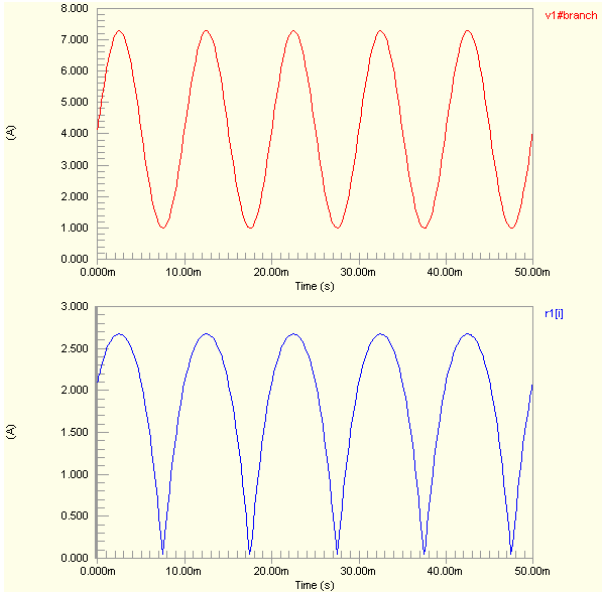


Consider the circuit in the image above. With respect to the ACOSHI component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN 0 OUT 0 ACOSHI
.
```

```
.
*Models and Subcircuit:
.SUBCKT ACOSHI 1 2 3 4
VX 1 2 0
BX 4 3 I=ACOSH(I(VX))
.ENDS ACOSHI
```

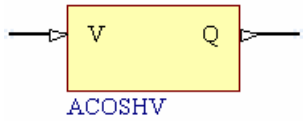
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Hyperbolic Arc Cosine of Voltage (Single-Ended Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ACOSHV

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @MODEL
```

Parameters (definable at component level)

None

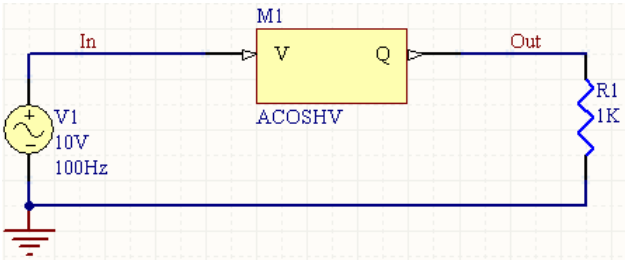
Notes

The content of the sub-circuit file (`ACOSHV.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Hyperbolic arc cosine of Voltage
.SUBCKT ACOSHV 1 2
BX 2 0 V=ACOSH(V(1))
.ENDS ACOSHV
```

The resulting voltage is the value expressed in radians.

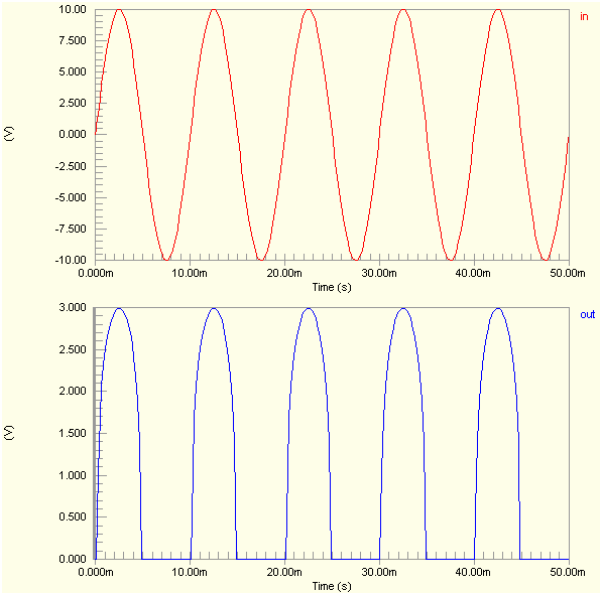
Examples



Consider the circuit in the image above. With respect to the ACOSHV component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN OUT ACOSHV
.
.
*Models and Subcircuit:
.SUBCKT ACOSHV 1 2
BX 2 0 V=ACOSH(V(1))
.ENDS ACOSHV
```

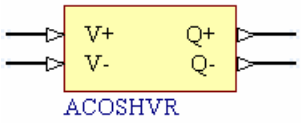
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Hyperbolic Arc Cosine of Voltage (Differential Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ACOSHVR

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

None

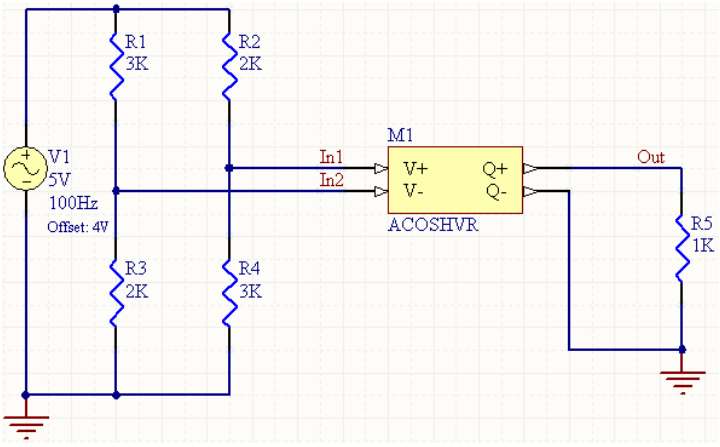
Notes

The content of the sub-circuit file (`ACOSHVR.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Hyperbolic arc cosine of Voltage
.SUBCKT ACOSHVR 1 2 3 4
BX 3 4 V=ACOSH(V(1,2))
.ENDS ACOSHVR
```

The resulting voltage is the value expressed in radians.

Examples

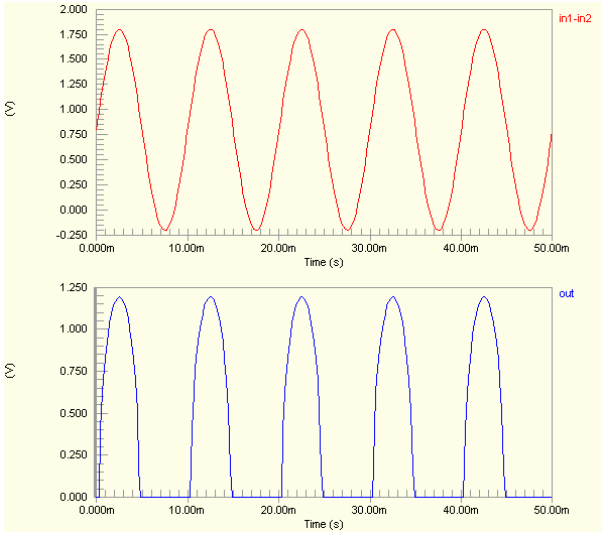


Consider the circuit in the image above. With respect to the ACOSHVR component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN1 IN2 OUT 0 ACOSHVR
.
.
*Models and Subcircuit:
.SUBCKT ACOSHVR 1 2 3 4
```

```
BX 3 4 V=ACOSH(V(1,2))
.ENDS ACOSHVR
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

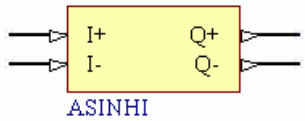


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Hyperbolic Arc Sine

Hyperbolic Arc Sine of Current



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ASINHI

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

None

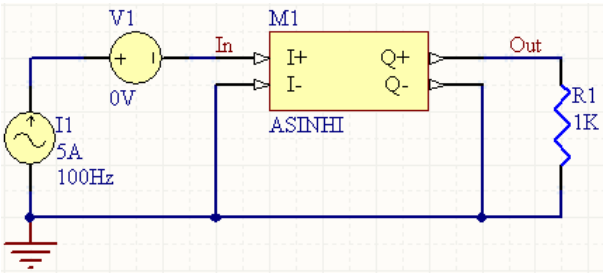
Notes

The content of the sub-circuit file (`ASINHI.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Hyperbolic arc sine of Current
.SUBCKT ASINHI 1 2 3 4
VX 1 2 0
BX 4 3 I=ASINH(I(VX))
.ENDS ASINHI
```

The resulting current is the value expressed in radians.

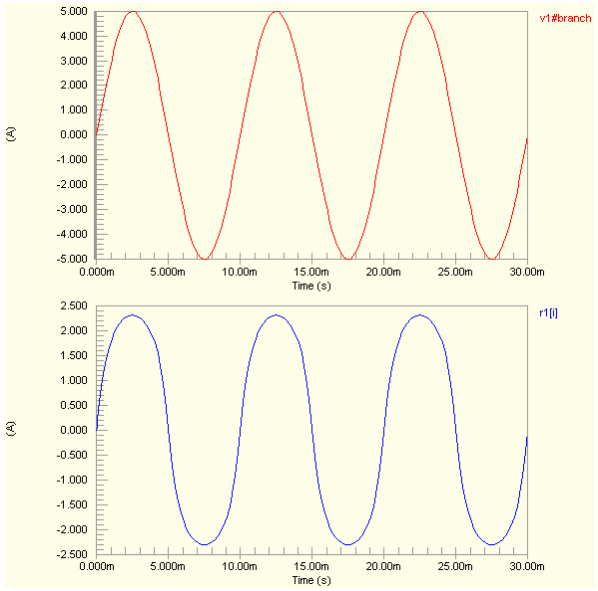
Examples



Consider the circuit in the image above. With respect to the ASINHI component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN 0 OUT 0 ASINHI
.
.
*Models and Subcircuit:
.SUBCKT ASINHI 1 2 3 4
VX 1 2 0
BX 4 3 I=ASINH(I(VX))
.ENDS ASINHI
```

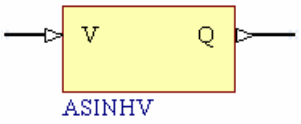
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Hyperbolic Arc Sine of Voltage (Single-Ended Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ASINHV

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @MODEL
```

Parameters (definable at component level)

None

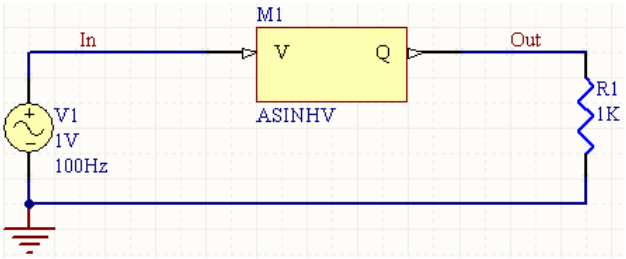
Notes

The content of the sub-circuit file (`ASINHV.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Hyperbolic arc sine of Voltage
.SUBCKT ASINHV 1 2
BX 2 0 V=ASINH(V(1))
.ENDS ASINHV
```

The resulting voltage is the value expressed in radians.

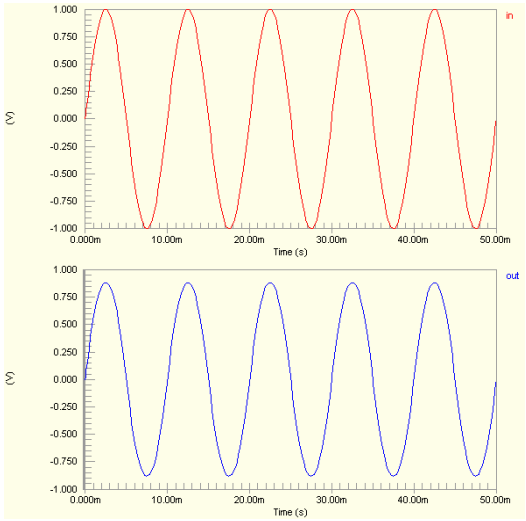
Examples



Consider the circuit in the image above. With respect to the ASINHV component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN OUT ASINHV
.
.
*Models and Subcircuit:
.SUBCKT ASINHV 1 2
BX 2 0 V=ASINH(V(1))
.ENDS ASINHV
```

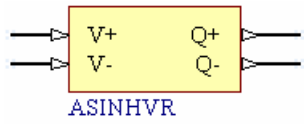
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Hyperbolic Arc Sine of Voltage (Differential Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ASINHVR

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

None

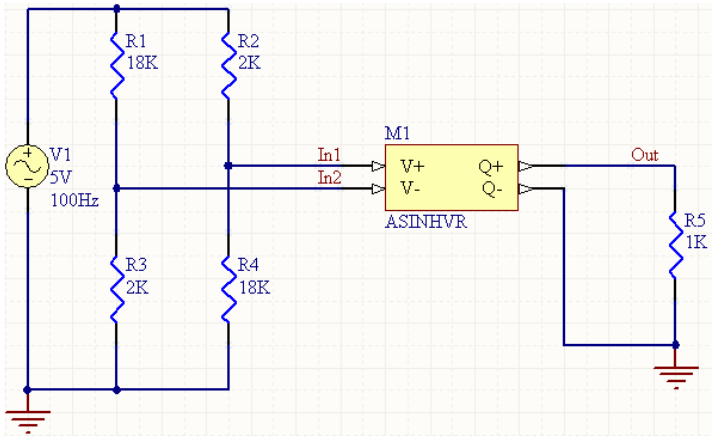
Notes

The content of the sub-circuit file (`ASINHVR.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Hyperbolic arc sine of Voltage
.SUBCKT ASINHVR 1 2 3 4
BX 3 4 V=ASINH(V(1,2))
.ENDS ASINHVR
```

The resulting voltage is the value expressed in radians.

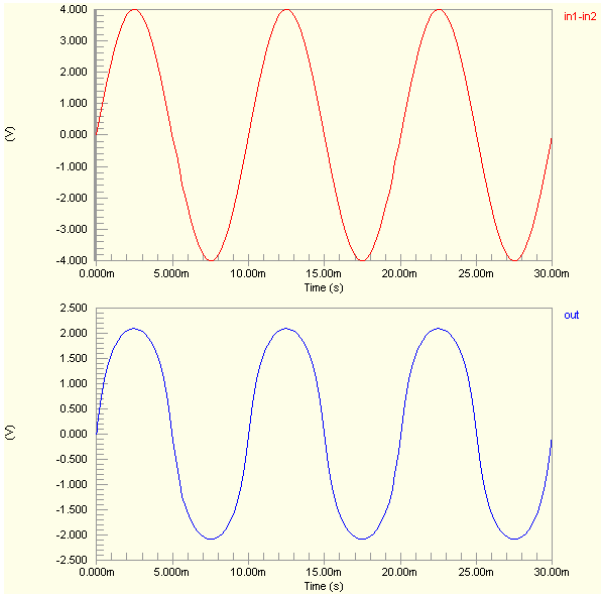
Examples



Consider the circuit in the image above. With respect to the ASINHVR component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN1 IN2 OUT 0 ASINHVR
.
.
*Models and Subcircuit:
.SUBCKT ASINHVR 1 2 3 4
BX 3 4 V=ASINH(V(1,2))
.ENDS ASINHVR
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

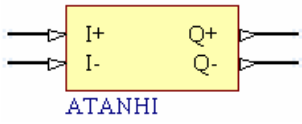


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Hyperbolic Arc Tangent

Hyperbolic Arc Tangent of Current



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ATANHI

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

None

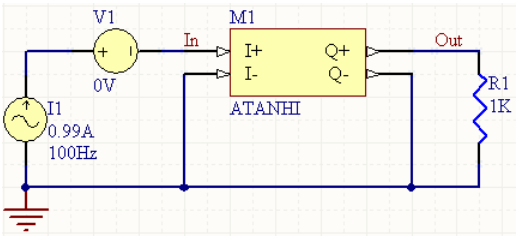
Notes

The content of the sub-circuit file (`ATANHI.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Hyperbolic arc tangent of Current
.SUBCKT ATANHI 1 2 3 4
VX 1 2 0
BX 4 3 I=ATANH(I(VX))
.ENDS ATANHI
```

The resulting current is the value expressed in radians.

Examples

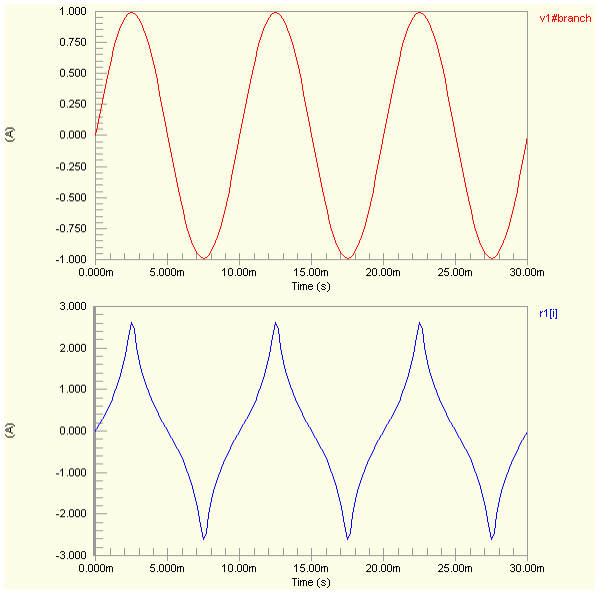


Consider the circuit in the image above. With respect to the ATANHI component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN 0 OUT 0 ATANHI
.
.
*Models and Subcircuit:
.SUBCKT ATANHI 1 2 3 4
VX 1 2 0
BX 4 3 I=ATANH(I(VX))
```

```
.ENDS ATANHI
```

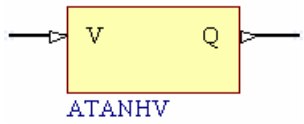
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Hyperbolic Arc Tangent of Voltage (Single-Ended Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ATANHV

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @MODEL
```

Parameters (definable at component level)

None

Notes

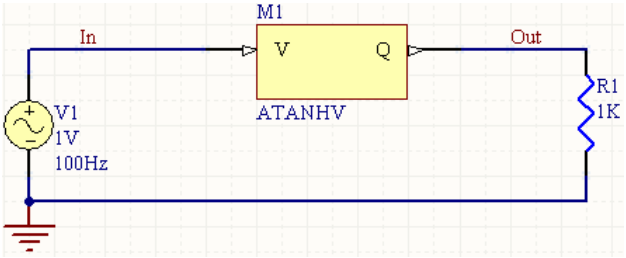
The content of the sub-circuit file (`ATANHV.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Hyperbolic arc tangent of Voltage
```

```
.SUBCKT ATANHV 1 2
BX 2 0 V=ATANH(V(1))
.ENDS ATANHV
```

The resulting voltage is the value expressed in radians.

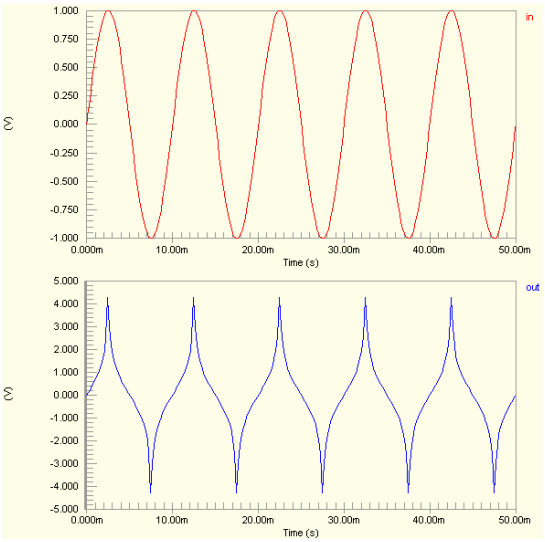
Examples



Consider the circuit in the image above. With respect to the ATANHV component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN OUT ATANHV
.
.
*Models and Subcircuit:
.SUBCKT ATANHV 1 2
BX 2 0 V=ATANH(V(1))
.ENDS ATANHV
```

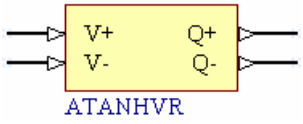
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Hyperbolic Arc Tangent of Voltage (Differential Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

ATANHVR

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

None

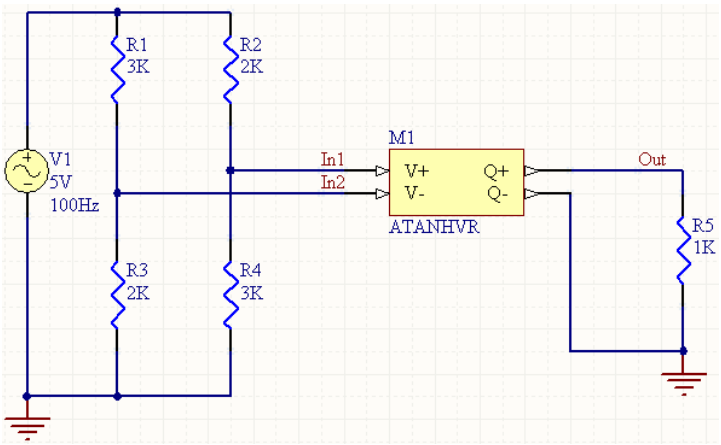
Notes

The content of the sub-circuit file (`ATANHVR.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Hyperbolic arc tangent of Voltage
.SUBCKT ATANHVR 1 2 3 4
BX 3 4 V=ATANH(V(1,2))
.ENDS ATANHVR
```

The resulting voltage is the value expressed in radians.

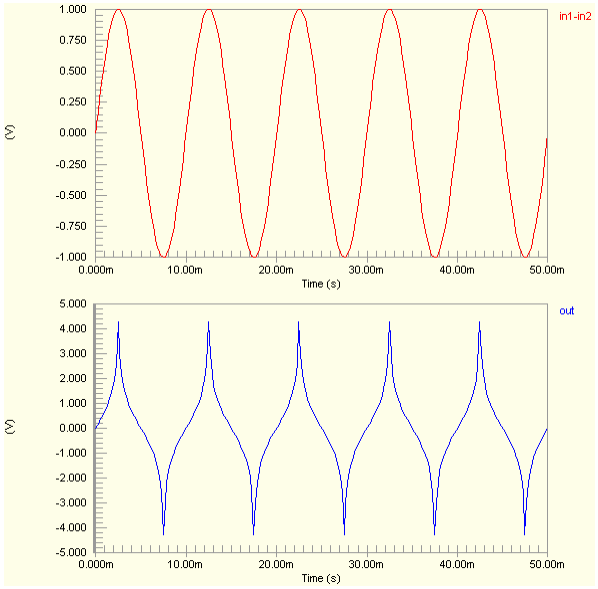
Examples



Consider the circuit in the image above. With respect to the ATANHVR component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN1 IN2 OUT 0 ATANHVR
.
.
*Models and Subcircuit:
.SUBCKT ATANHVR 1 2 3 4
BX 3 4 V=ATANH(V(1,2))
.ENDS ATANHVR
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

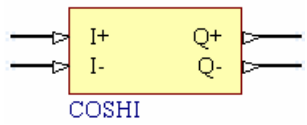


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Hyperbolic Cosine

Hyperbolic Cosine of Current



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

COSHI

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

None

Notes

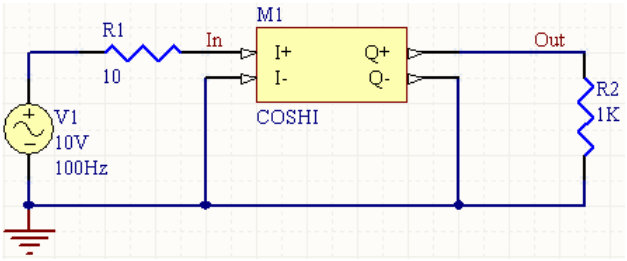
The content of the sub-circuit file (`COSHI.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Hyperbolic cosine of Current
.SUBCKT COSHI 1 2 3 4
```

```
VX 1 2 0
BX 4 3 I=COSH(I(VX))
.ENDS COSHI
```

The resulting current is the value expressed in radians.

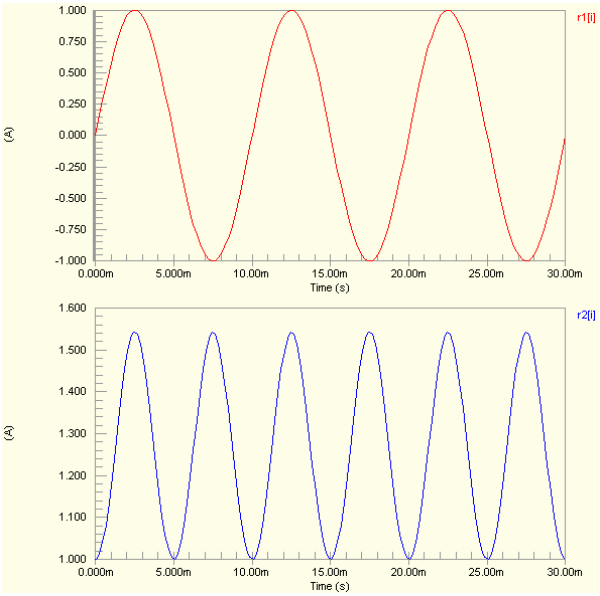
Examples



Consider the circuit in the image above. With respect to the COSHI component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN 0 OUT 0 COSHI
.
.
*Models and Subcircuit:
.SUBCKT COSHI 1 2 3 4
VX 1 2 0
BX 4 3 I=COSH(I(VX))
.ENDS COSHI
```

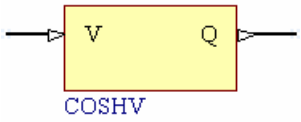
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Hyperbolic Cosine of Voltage (Single-Ended Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

COSHV

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @MODEL
```

Parameters (definable at component level)

None

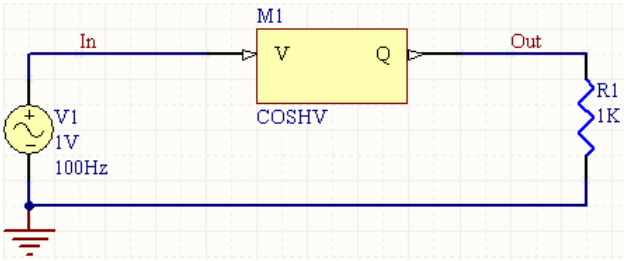
Notes

The content of the sub-circuit file (`COSHV.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Hyperbolic cosine of Voltage
.SUBCKT COSHV 1 2
BX 2 0 V=COSH(V(1))
.ENDS COSHV
```

The resulting voltage is the value expressed in radians.

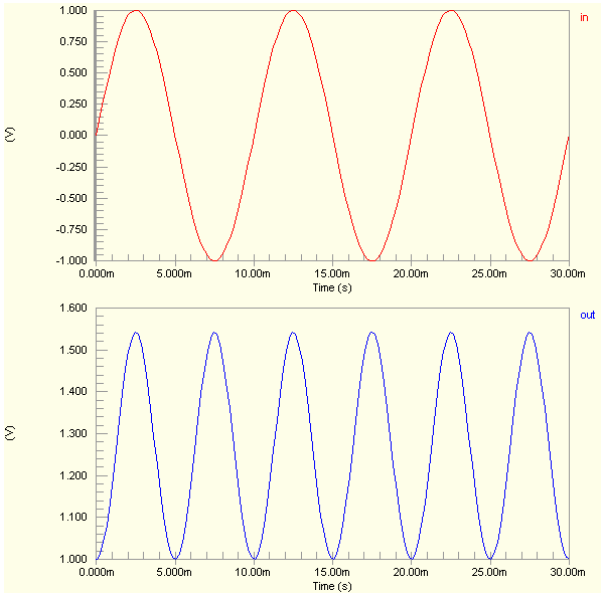
Examples



Consider the circuit in the image above. With respect to the COSHV component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN OUT COSHV
.
.
*Models and Subcircuit:
.SUBCKT COSHV 1 2
BX 2 0 V=COSH(V(1))
.ENDS COSHV
```

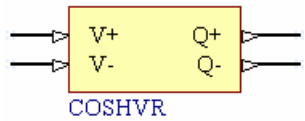
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Hyperbolic Cosine of Voltage (Differential Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

COSHVR

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

None

Notes

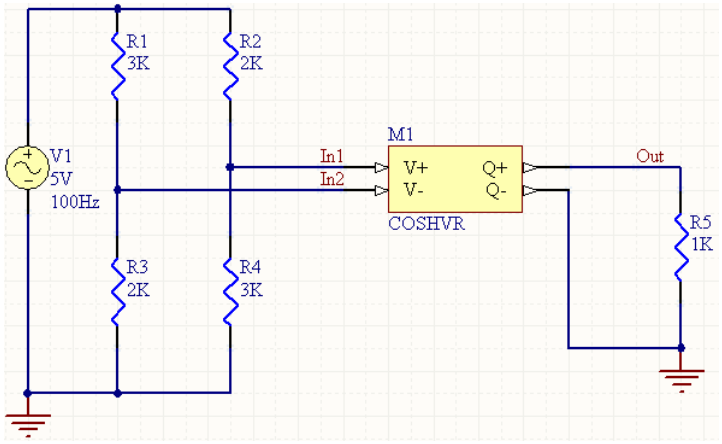
The content of the sub-circuit file (`COSHVR.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Hyperbolic cosine of Voltage
.SUBCKT COSHVR 1 2 3 4
BX 3 4 V=COSH(V(1,2))
```

```
.ENDS COSHVR
```

The resulting voltage is the value expressed in radians.

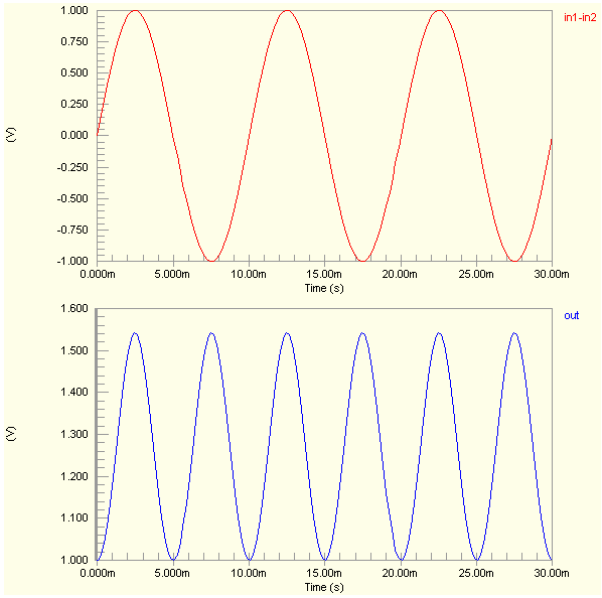
Examples



Consider the circuit in the image above. With respect to the COSHVR component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN1 IN2 OUT 0 COSHVR
.
.
*Models and Subcircuit:
.SUBCKT COSHVR 1 2 3 4
BX 3 4 V=COSH(V(1,2))
.ENDS COSHVR
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

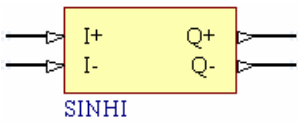


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Hyperbolic Sine

Hyperbolic Sine of Current



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

SINHI

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

None

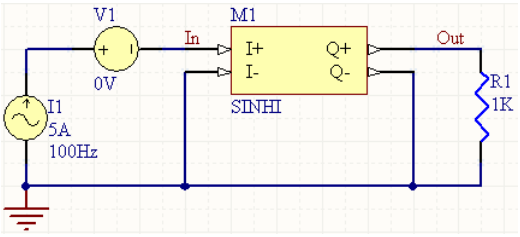
Notes

The content of the sub-circuit file (`SINHI.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Hyperbolic sine of Current
.SUBCKT SINHI 1 2 3 4
VX 1 2 0
BX 4 3 I=SINH(I(VX))
.ENDS SINHI
```

The resulting current is the value expressed in radians.

Examples

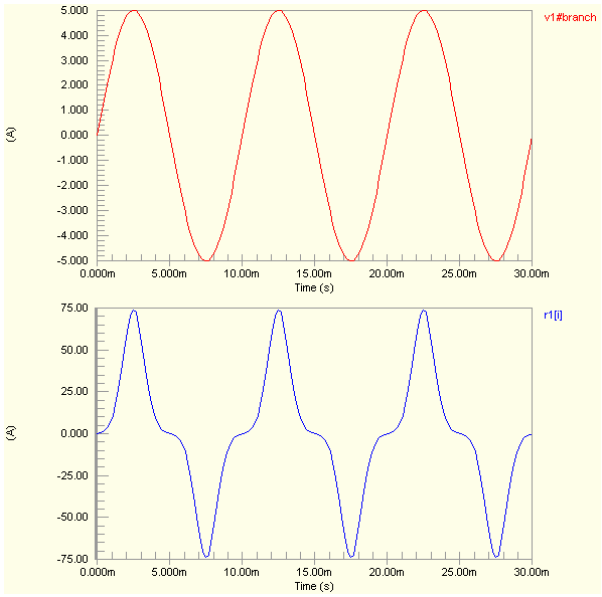


Consider the circuit in the image above. With respect to the SINHI component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN 0 OUT 0 SINHI
.
.
*Models and Subcircuit:
.SUBCKT SINHI 1 2 3 4
VX 1 2 0
BX 4 3 I=SINH(I(VX))
```

```
.ENDS SINHI
```

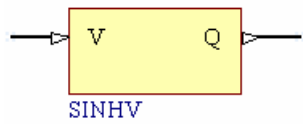
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Hyperbolic Sine of Voltage (Single-Ended Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

SINHV

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @MODEL
```

Parameters (definable at component level)

None

Notes

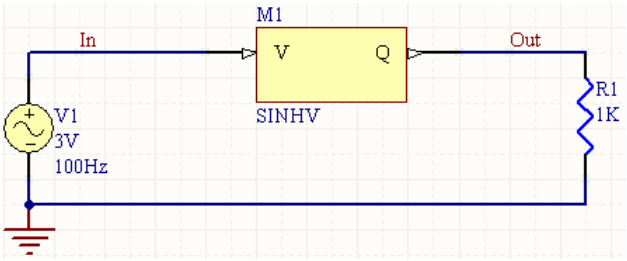
The content of the sub-circuit file (`SINHV.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Hyperbolic sine of Voltage
```

```
.SUBCKT SINHV 1 2
BX 2 0 V=SINH(V(1))
.ENDS SINHV
```

The resulting voltage is the value expressed in radians.

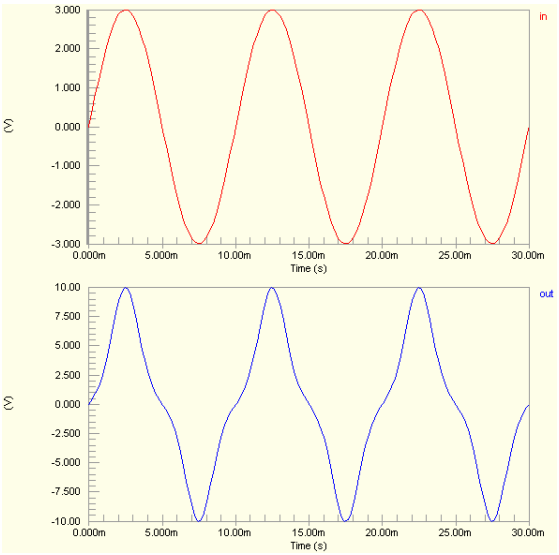
Examples



Consider the circuit in the image above. With respect to the SINHV component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN OUT SINHV
.
.
*Models and Subcircuit:
.SUBCKT SINHV 1 2
BX 2 0 V=SINH(V(1))
.ENDS SINHV
```

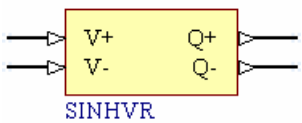
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Hyperbolic Sine of Voltage (Differential Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

SINHVR

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

None

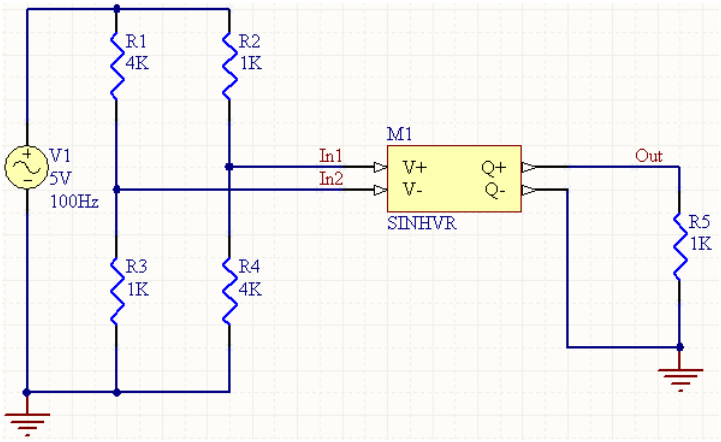
Notes

The content of the sub-circuit file (`SINHVR.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Hyperbolic sine of Voltage
.SUBCKT SINHVR 1 2 3 4
BX 3 4 V=SINH(V(1,2))
.ENDS SINHVR
```

The resulting voltage is the value expressed in radians.

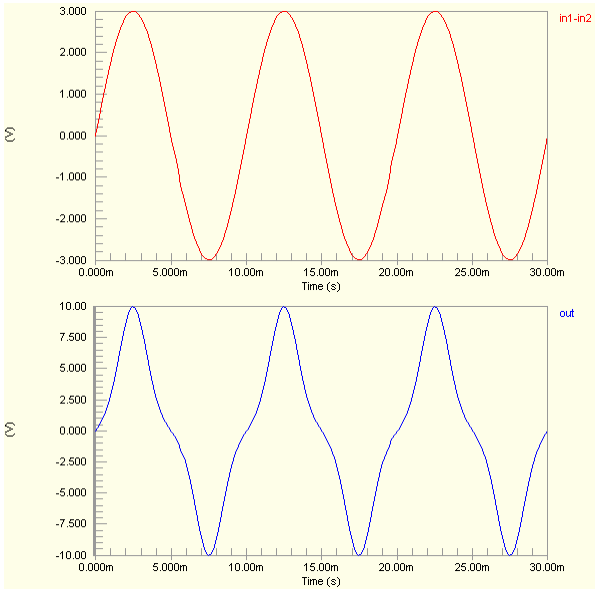
Examples



Consider the circuit in the image above. With respect to the SINHVR component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN1 IN2 OUT 0 SINHVR
.
.
*Models and Subcircuit:
.SUBCKT SINHVR 1 2 3 4
BX 3 4 V=SINH(V(1,2))
.ENDS SINHVR
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

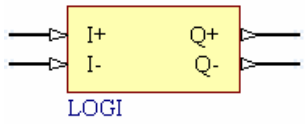


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Logarithm (Base 10)

Logarithm of Current



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

LOGI

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

None

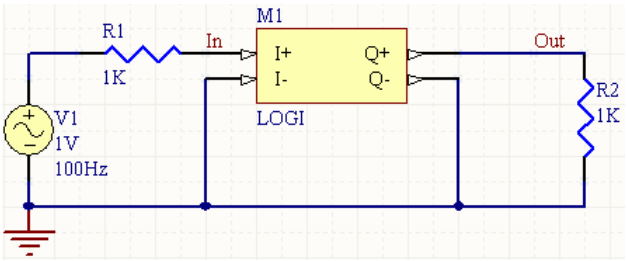
Notes

The content of the sub-circuit file (`LOGI.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Logarithm of Current
.SUBCKT LOGI 1 2 3 4
```

```
VX 1 2 0
BX 4 3 I=LOG(I(VX))
.ENDS LOGI
```

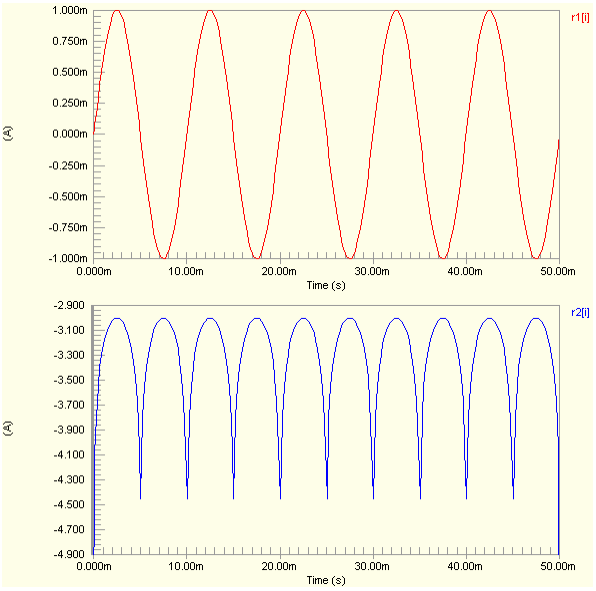
Examples



Consider the circuit in the image above. With respect to the LOGI component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN 0 OUT 0 LOGI
.
.
*Models and Subcircuit:
.SUBCKT LOGI 1 2 3 4
VX 1 2 0
BX 4 3 I=LOG(I(VX))
.ENDS LOGI
```

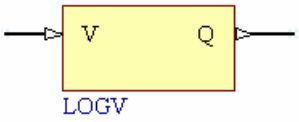
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Logarithm of Voltage (Single-Ended Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

LOGV

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @MODEL
```

Parameters (definable at component level)

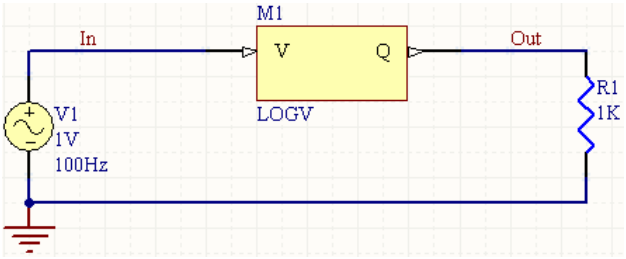
None

Notes

The content of the sub-circuit file (`LOGV.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Logarithm of Voltage
.SUBCKT LOGV 1 2
BX 2 0 V=LOG(V(1))
.ENDS LOGV
```

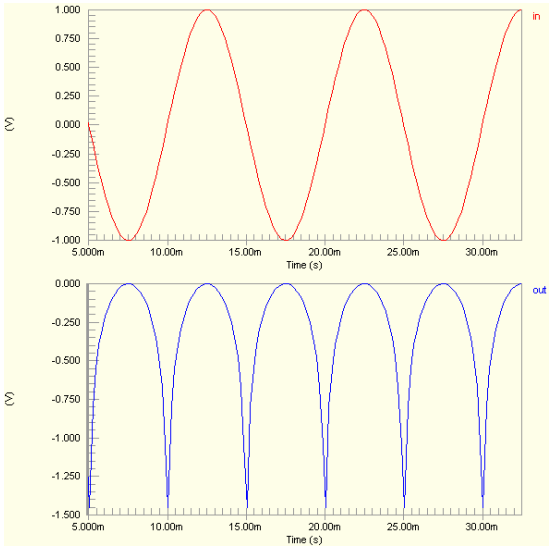
Examples



Consider the circuit in the image above. With respect to the LOGV component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN OUT LOGV
.
.
*Models and Subcircuit:
.SUBCKT LOGV 1 2
BX 2 0 V=LOG(V(1))
.ENDS LOGV
```

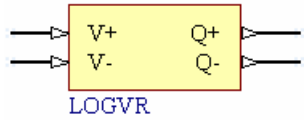
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Logarithm of Voltage (Differential Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

LOGVR

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

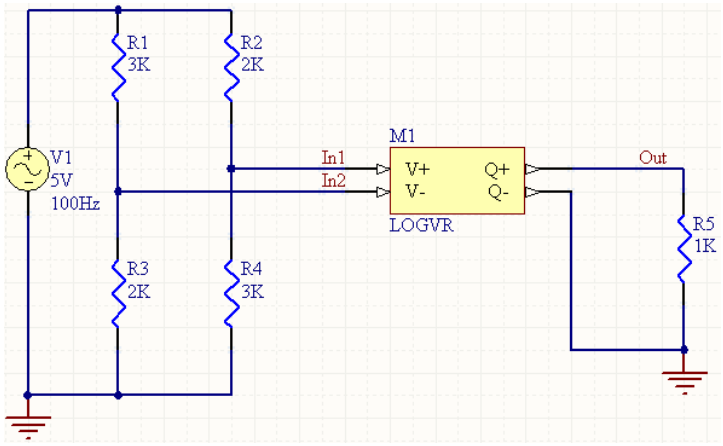
None

Notes

The content of the sub-circuit file (`LOGVR.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Logarithm of Voltage
.SUBCKT LOGVR 1 2 3 4
BX 3 4 V=LOG(V(1,2))
.ENDS LOGVR
```

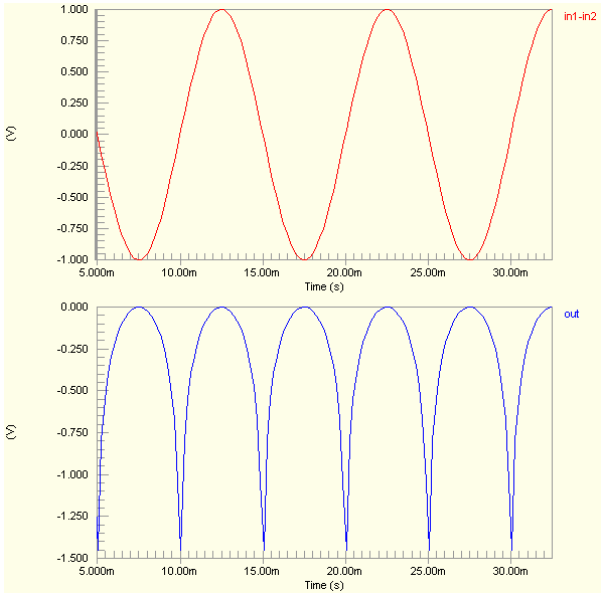
Examples



Consider the circuit in the image above. With respect to the LOGVR component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN1 IN2 OUT 0 LOGVR
.
.
*Models and Subcircuit:
.SUBCKT LOGVR 1 2 3 4
BX 3 4 V=LOG(V(1,2))
.ENDS LOGVR
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

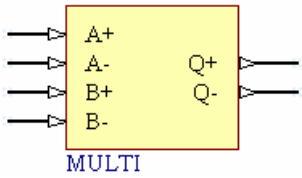


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Multiplication

Multiplication of Currents



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

MULTI

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 %5 %6 @MODEL
```

Parameters (definable at component level)

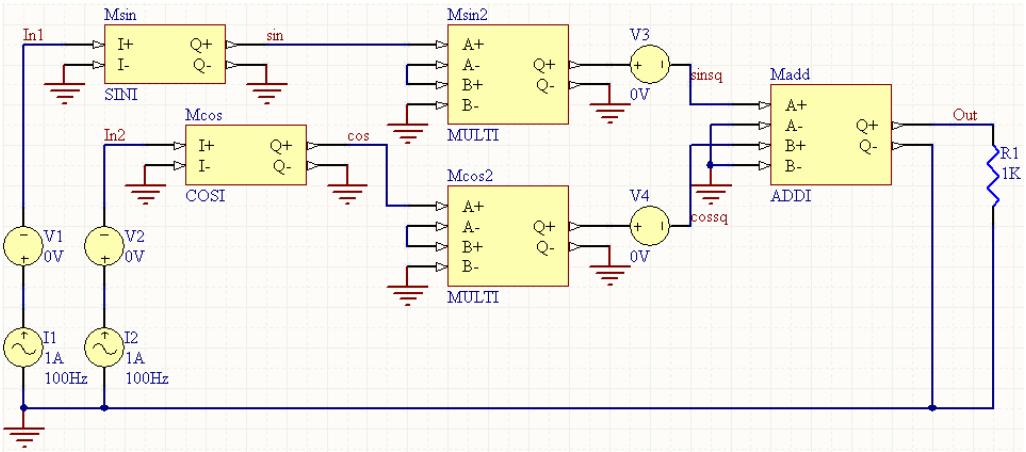
None

Notes

The content of the sub-circuit file (`MULTI.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Multiply Currents
.SUBCKT MULTI 1 2 3 4 5 6
VA 1 2 0
VB 3 4 0
BX 6 5 I=I(VA)*I(VB)
.ENDS MULTI
```

Examples



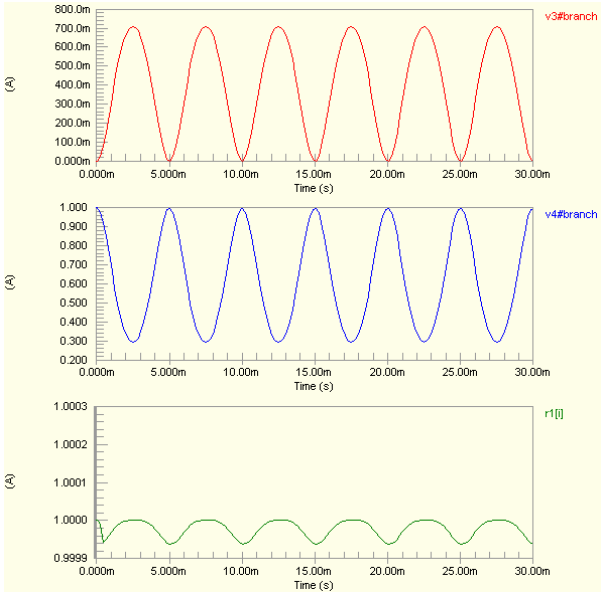
Consider the circuit in the image above, which uses math function components to implement the trigonometric base equation

$$Sin^2(I) + Cos^2(I) = 1$$

With respect to the MULTI components, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XMcos2 COS NetMcos2_2 NetMcos2_2 0 NetMcos2_5 0 MULTI
XMsin2 SIN NetMsin2_2 NetMsin2_2 0 NetMsin2_5 0 MULTI
.
.
*Models and Subcircuit:
.SUBCKT MULTI 1 2 3 4 5 6
VA 1 2 0
VB 3 4 0
BX 6 5 I=I(VA)*I(VB)
.ENDS MULTI
```

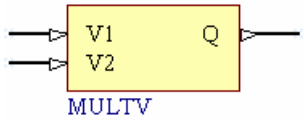
The effect of the functions can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Multiplication of Voltages (Single-Ended Inputs)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

MULTV

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 @MODEL
```

Parameters (definable at component level)

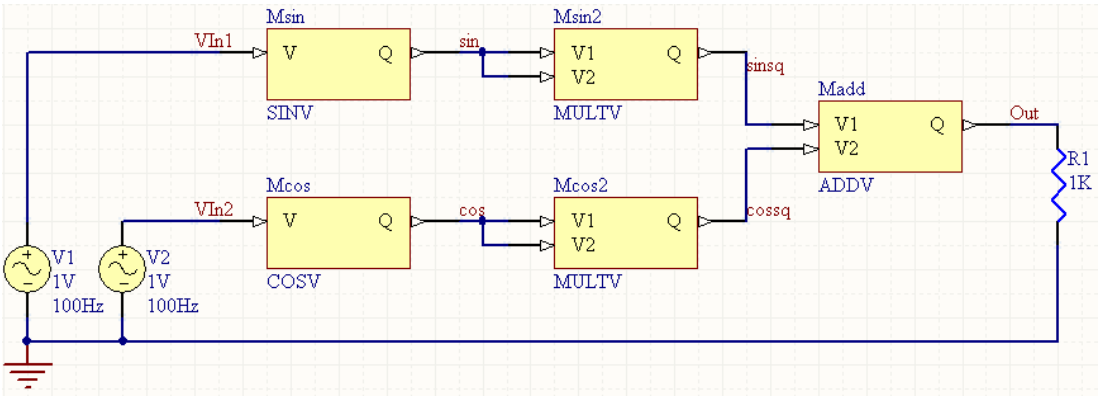
None

Notes

The content of the sub-circuit file (`MULTV.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Multiply Voltages
.SUBCKT MULTV 1 2 3
BX 3 0 V=V(1)*V(2)
.ENDS MULTV
```

Examples



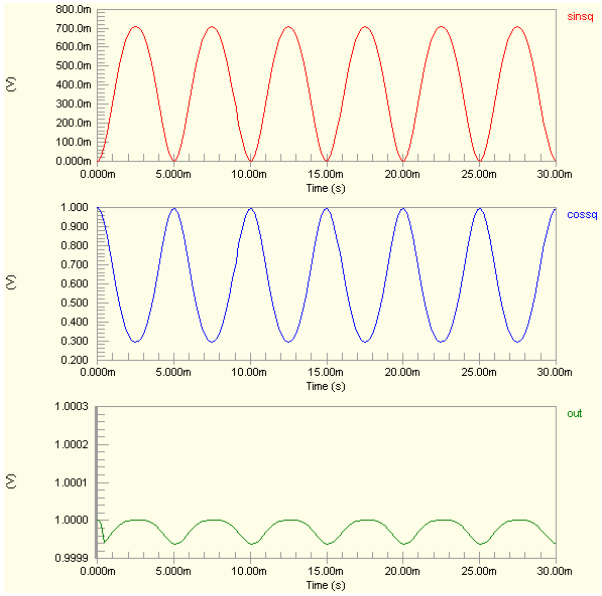
Consider the circuit in the image above, which uses math function components to implement the trigonometric base equation

$$Sin^2(v) + Cos^2(v) = 1$$

With respect to the MULTV components, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XMcos2 COS COS COSSQ MULTV
XMsin2 SIN SIN SINSQ MULTV
.
.
*Models and Subcircuit:
.SUBCKT MULTV 1 2 3
BX 3 0 V=V(1)*V(2)
.ENDS MULTV
```

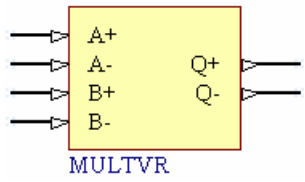
The effect of the functions can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Multiplication of Voltages (Differential Inputs)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

MULTVR

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 %5 %6 @MODEL
```

Parameters (definable at component level)

None

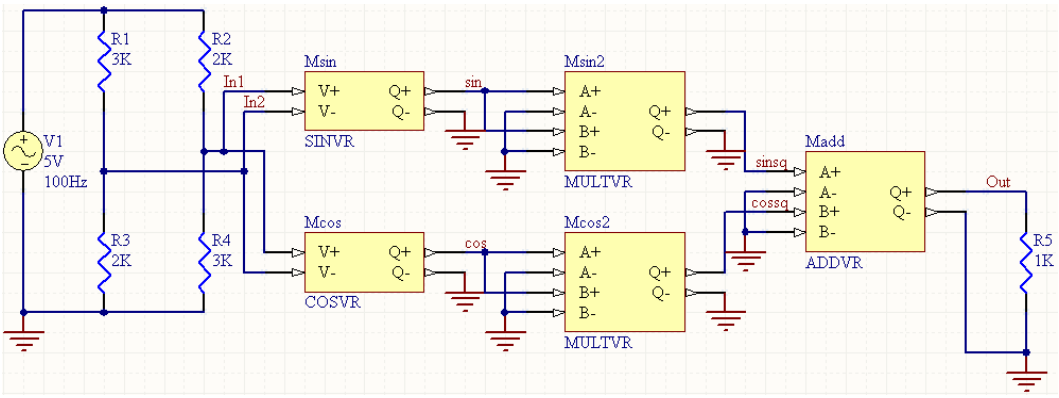
Notes

The content of the sub-circuit file (`MULTVR.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Multiply Voltages
.SUBCKT MULTVR 1 2 3 4 5 6
```

```
BX 5 6 V=V(1,2)*V(3,4)
.ENDS MULTVR
```

Examples



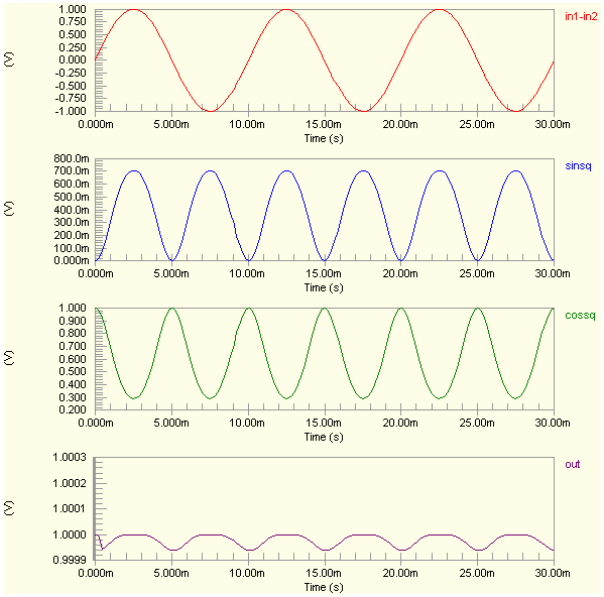
Consider the circuit in the image above, which uses math function components to implement the trigonometric base equation

$Sin^2(v) + Cos^2(v) = 1$

With respect to the MULTVR components, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XMCos2 COS 0 COS 0 COSSQ 0 MULTVR
XMSin2 SIN 0 SIN 0 SINSQ 0 MULTVR
.
.
*Models and Subcircuit:
.SUBCKT MULTVR 1 2 3 4 5 6
BX 5 6 V=V(1,2)*V(3,4)
.ENDS MULTVR
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



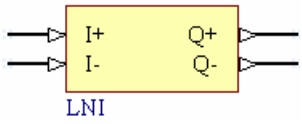
In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`

- **Transient Max Step Time** – set to `200.0u`.

Natural Logarithm (Base e)

Natural Logarithm of Current



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

LNI

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

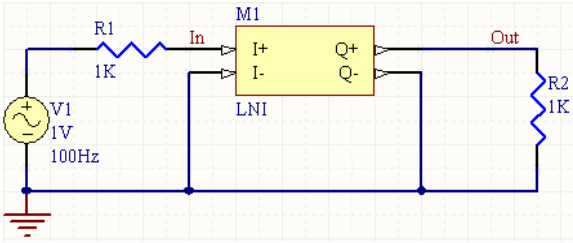
None

Notes

The content of the sub-circuit file (`LNI.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Natural logarithm of Current
.SUBCKT LNI 1 2 3 4
VX 1 2 0
BX 4 3 I=LN(I(VX))
.ENDS LNI
```

Examples

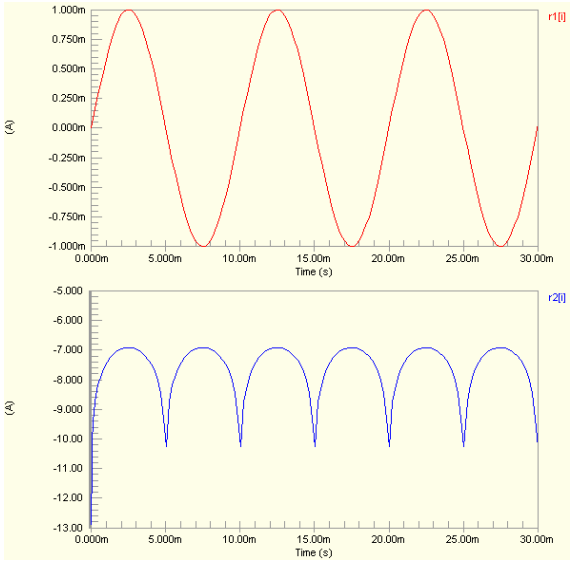


Consider the circuit in the image above. With respect to the LOGI component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN 0 OUT 0 LNI
.
.
*Models and Subcircuit:
.SUBCKT LNI 1 2 3 4
VX 1 2 0
BX 4 3 I=LN(I(VX))
```

```
.ENDS LNI
```

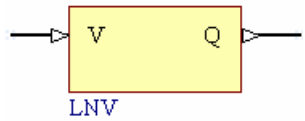
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Natural Logarithm of Voltage (Single-Ended Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

LNV

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @MODEL
```

Parameters (definable at component level)

None

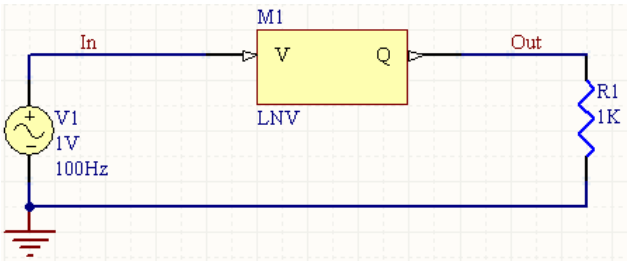
Notes

The content of the sub-circuit file (`LNV.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Natural logarithm of Voltage
.SUBCKT LNV 1 2
```

```
BX 2 0 V=LN(V(1))
.ENDS LNV
```

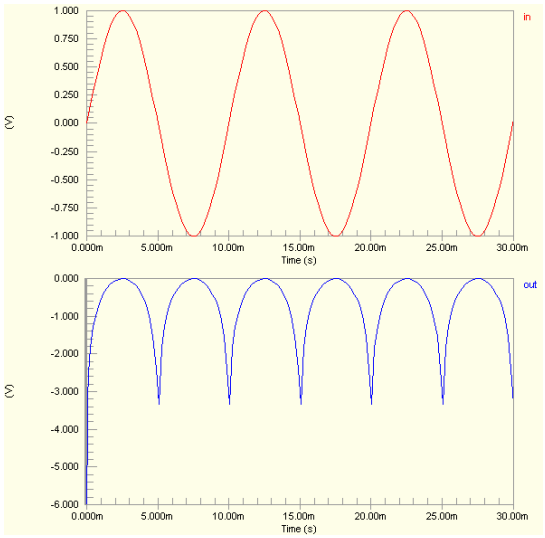
Examples



Consider the circuit in the image above. With respect to the LNV component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN OUT LNV
.
.
*Models and Subcircuit:
.SUBCKT LNV 1 2
BX 2 0 V=LN(V(1))
.ENDS LNV
```

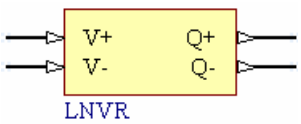
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Natural Logarithm of Voltage (Differential Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

LNVR

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

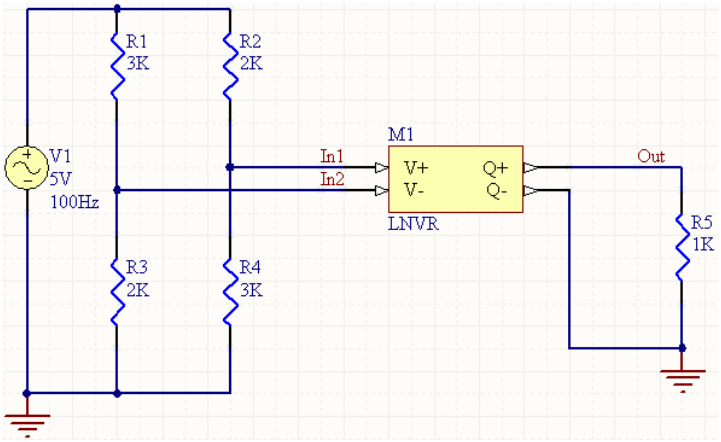
None

Notes

The content of the sub-circuit file (`LNVR.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Natural logarithm of Voltage
.SUBCKT LNVR 1 2 3 4
BX 3 4 V=LN(V(1,2))
.ENDS LNVR
```

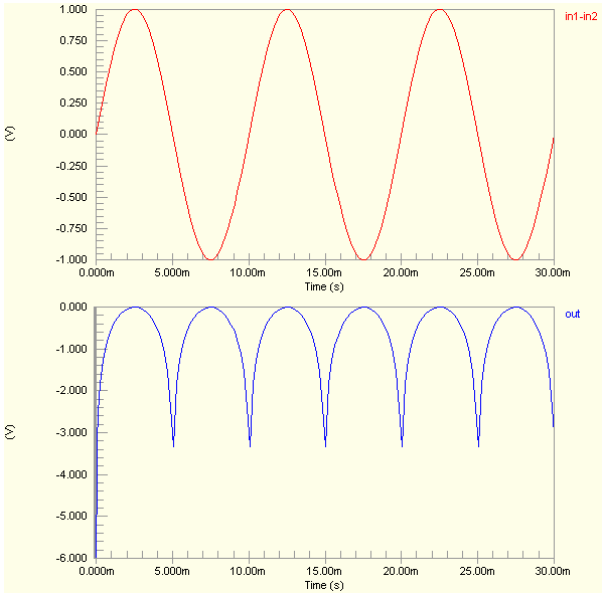
Examples



Consider the circuit in the image above. With respect to the LNVR component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN1 IN2 OUT 0 LNVR
.
.
*Models and Subcircuit:
.SUBCKT LNVR 1 2 3 4
BX 3 4 V=LN(V(1,2))
.ENDS LNVR
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

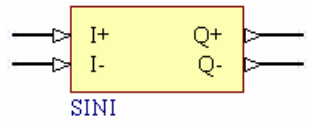


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Sine

Sine of Current



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

SINI

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

None

Notes

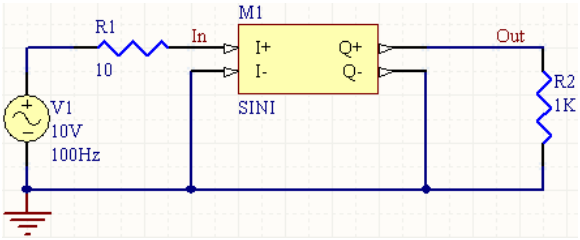
The content of the sub-circuit file (`SINI.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Sine of Current
.SUBCKT SINI 1 2 3 4
```

```
VX 1 2 0
BX 4 3 I=SIN(I(VX))
.ENDS SINI
```

The resulting current is the value expressed in radians.

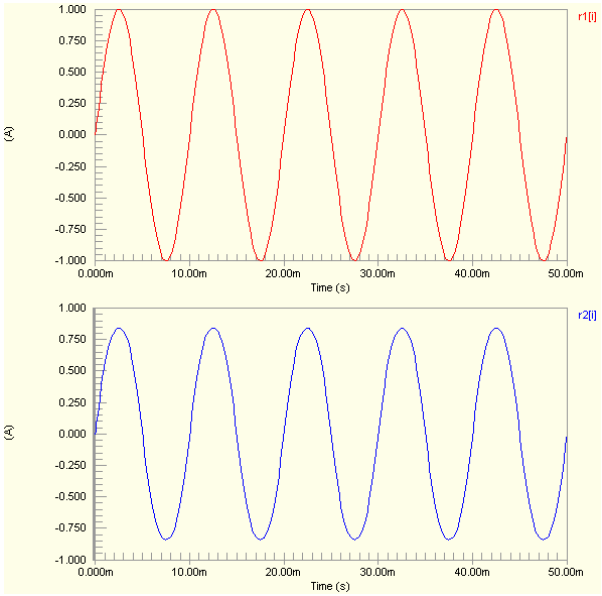
Examples



Consider the circuit in the image above. With respect to the SINI component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN 0 OUT 0 SINI
.
.
*Models and Subcircuit:
.SUBCKT SINI 1 2 3 4
VX 1 2 0
BX 4 3 I=SIN(I(VX))
.ENDS SINI
```

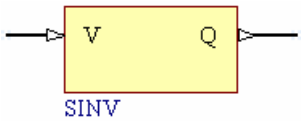
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Sine of Voltage (Single-Ended Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

SINV

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @MODEL
```

Parameters (definable at component level)

None

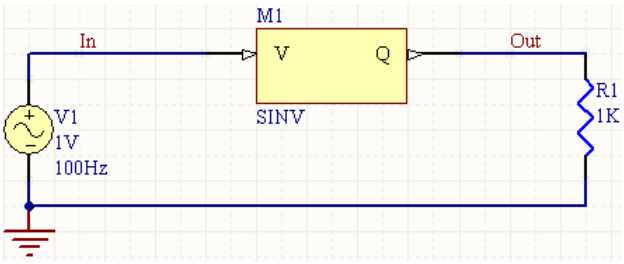
Notes

The content of the sub-circuit file (`SINV.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Sine of Voltage
.SUBCKT SINV 1 2
BX 2 0 V=SIN(V(1))
.ENDS SINV
```

The resulting voltage is the value expressed in radians.

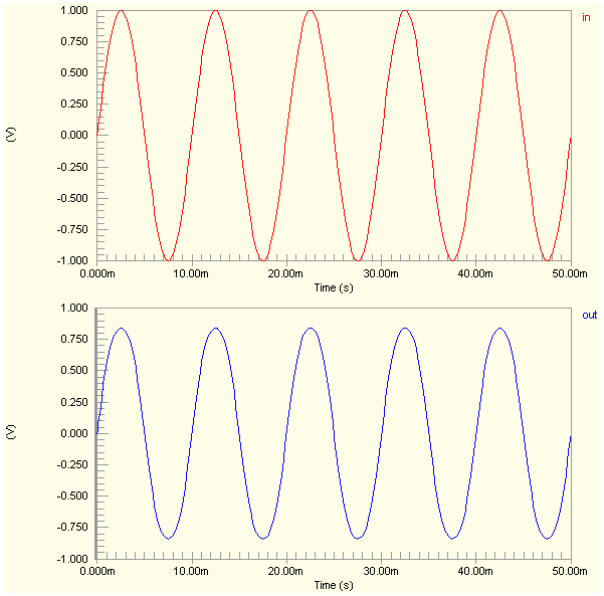
Examples



Consider the circuit in the image above. With respect to the SINV component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN OUT SINV
.
.
*Models and Subcircuit:
.SUBCKT SINV 1 2
BX 2 0 V=SIN(V(1))
.ENDS SINV
```

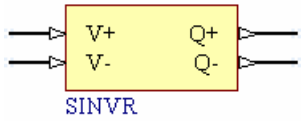
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Sine of Voltage (Differential Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

SINVR

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

None

Notes

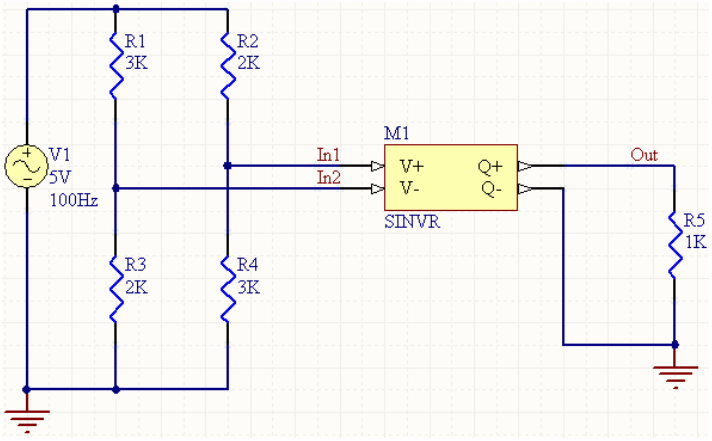
The content of the sub-circuit file (`SINVR.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Sine of Voltage
.SUBCKT SINVR 1 2 3 4
BX 3 4 V=SIN(V(1,2))
```

```
.ENDS SINVR
```

The resulting voltage is the value expressed in radians.

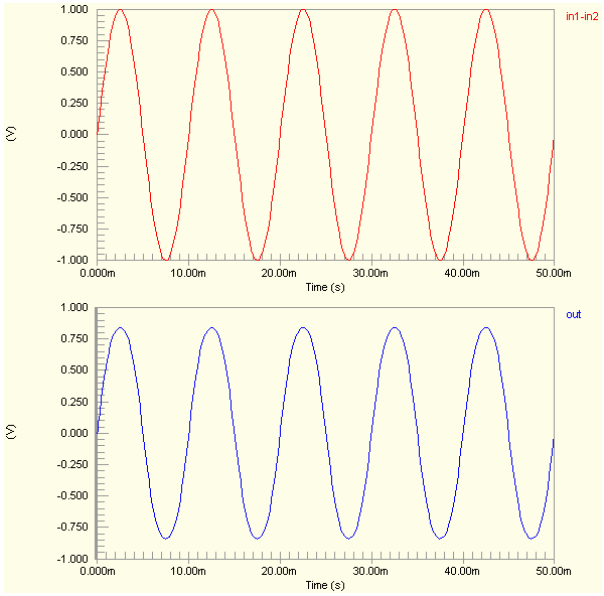
Examples



Consider the circuit in the image above. With respect to the SINVR component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN1 IN2 OUT 0 SINVR
.
.
*Models and Subcircuit:
.SUBCKT SINVR 1 2 3 4
BX 3 4 V=SIN(V(1,2))
.ENDS SINVR
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

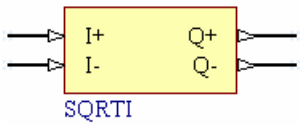


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Square Root

Square Root of Current



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

SQRTI

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

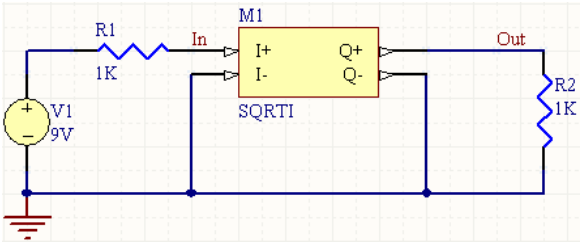
None

Notes

The content of the sub-circuit file (`SQRTI.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Square root of Current
.SUBCKT SQRTI 1 2 3 4
VX 1 2 0
BX 4 3 I=SQRT(I(VX))
.ENDS SQRTI
```

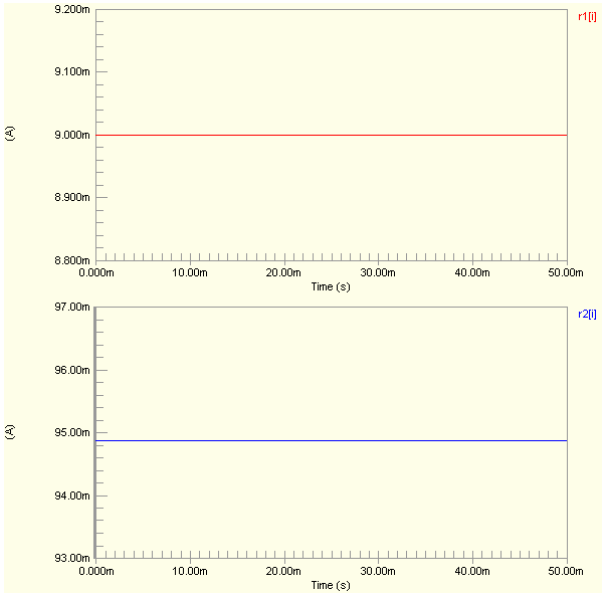
Examples



Consider the circuit in the image above. With respect to the SQRTI component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN 0 OUT 0 SQRTI
.
.
*Models and Subcircuit:
.SUBCKT SQRTI 1 2 3 4
VX 1 2 0
BX 4 3 I=SQRT(I(VX))
.ENDS SQRTI
```

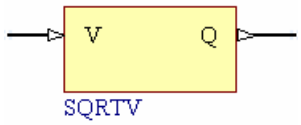
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Square Root of Voltage (Single-Ended Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

SQRTV

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @MODEL
```

Parameters (definable at component level)

None

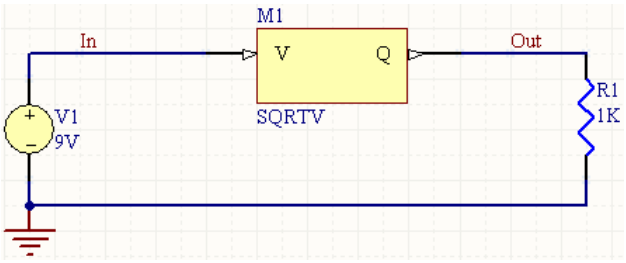
Notes

The content of the sub-circuit file (`SQRTV.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Square root of Voltage
.SUBCKT SQRTV 1 2
```

```
BX 2 0 V=SQRT(V(1))
.ENDS SQRTV
```

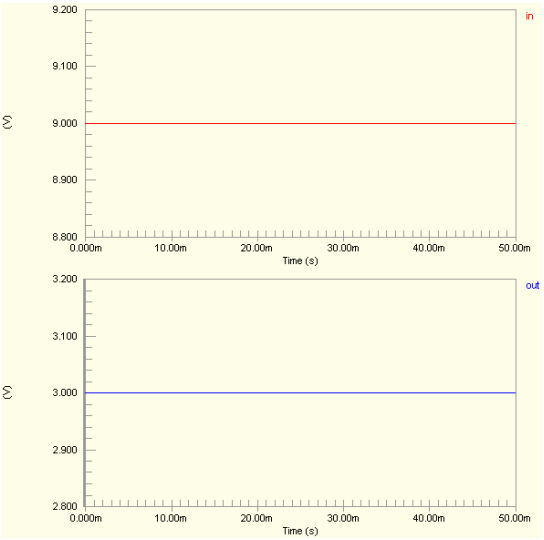
Examples



Consider the circuit in the image above. With respect to the SQRTV component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN OUT SQRTV
.
.
*Models and Subcircuit:
.SUBCKT SQRTV 1 2
BX 2 0 V=SQRT(V(1))
.ENDS SQRTV
```

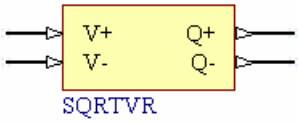
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Square Root of Voltage (Differential Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

SQRTVR

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

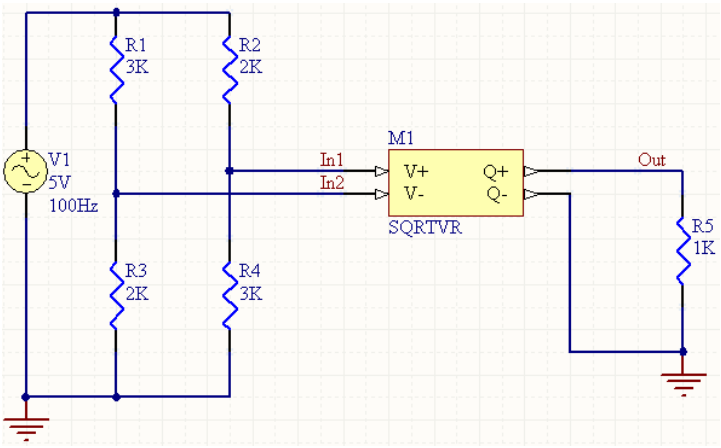
None

Notes

The content of the sub-circuit file (`SQRTVR.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Square root of Voltage
.SUBCKT SQRTVR 1 2 3 4
BX 3 4 V=SQRT(V(1,2))
.ENDS SQRTVR
```

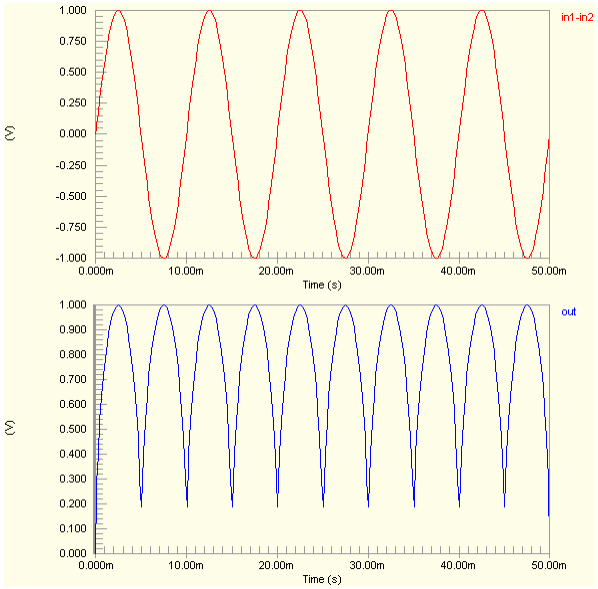
Examples



Consider the circuit in the image above. With respect to the SQRTVR component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN1 IN2 OUT 0 SQRTVR
.
.
*Models and Subcircuit:
.SUBCKT SQRTVR 1 2 3 4
BX 3 4 V=SQRT(V(1,2))
.ENDS SQRTVR
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

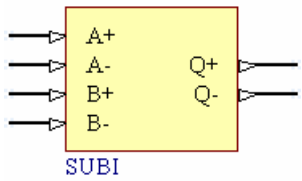


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Subtraction

Subtraction of Currents



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

SUBI

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 %5 %6 @MODEL
```

Parameters (definable at component level)

None

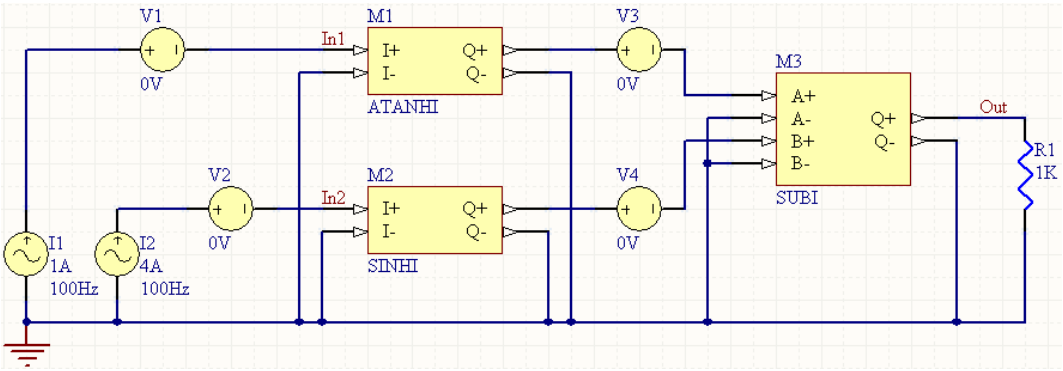
Notes

The content of the sub-circuit file (`SUBI.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Subtract Currents
```

```
.SUBCKT SUBI 1 2 3 4 5 6
VA 1 2 0
VB 3 4 0
BX 6 5 I=I(VA)-I(VB)
.ENDS SUBI
```

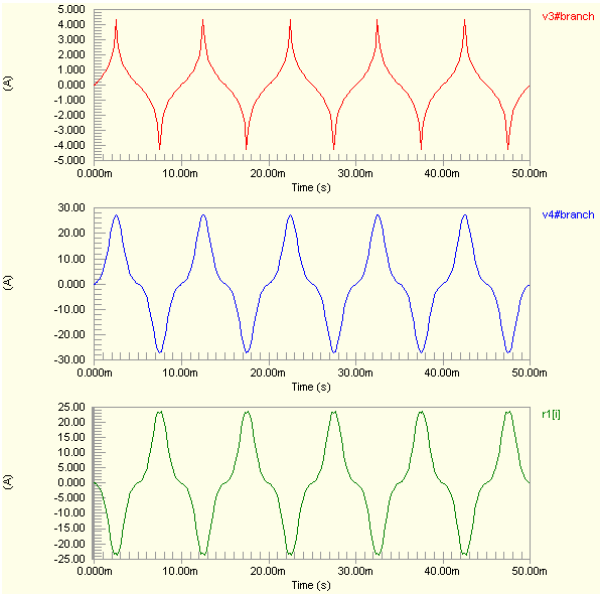
Examples



Consider the circuit in the image above. With respect to the SUBI component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM3 NetM3_1 0 NetM3_3 0 OUT 0 SUBI
.
.
*Models and Subcircuit:
.SUBCKT SUBI 1 2 3 4 5 6
VA 1 2 0
VB 3 4 0
BX 6 5 I=I(VA)-I(VB)
.ENDS SUBI
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.

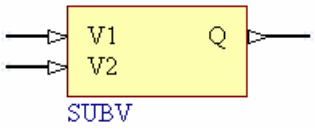


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`

- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Subtraction of Voltages (Single-Ended Inputs)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

SUBV

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 @MODEL
```

Parameters (definable at component level)

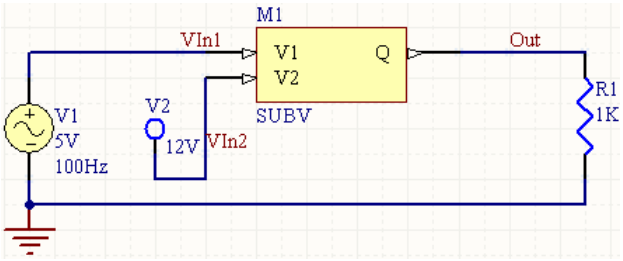
None

Notes

The content of the sub-circuit file (`SUBV.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Subrtact Voltages
.SUBCKT SUBV 1 2 3
BX 3 0 V=V(1)-V(2)
.ENDS SUBV
```

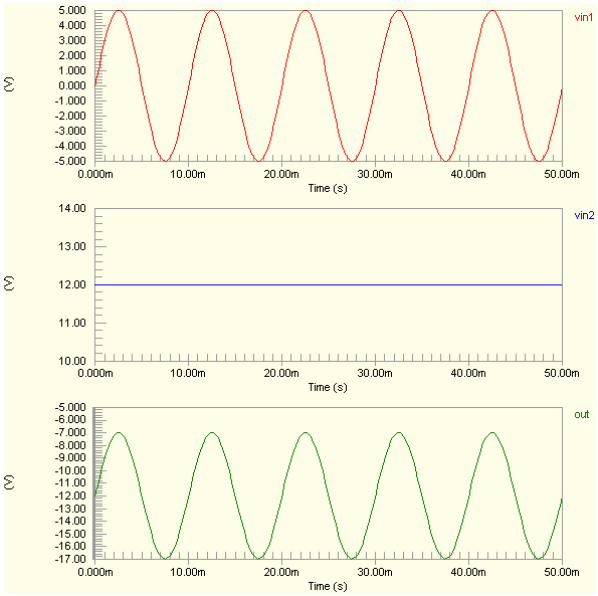
Examples



Consider the circuit in the image above. With respect to the SUBV component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 VIN1 VIN2 OUT SUBV
.
.
*Models and Subcircuit:
.SUBCKT SUBV 1 2 3
BX 3 0 V=V(1)-V(2)
.ENDS SUBV
```

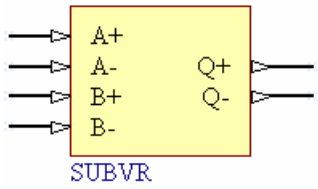
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Subtraction of Voltages (Differential Inputs)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

SUBVR

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 %5 %6 @MODEL
```

Parameters (definable at component level)

None

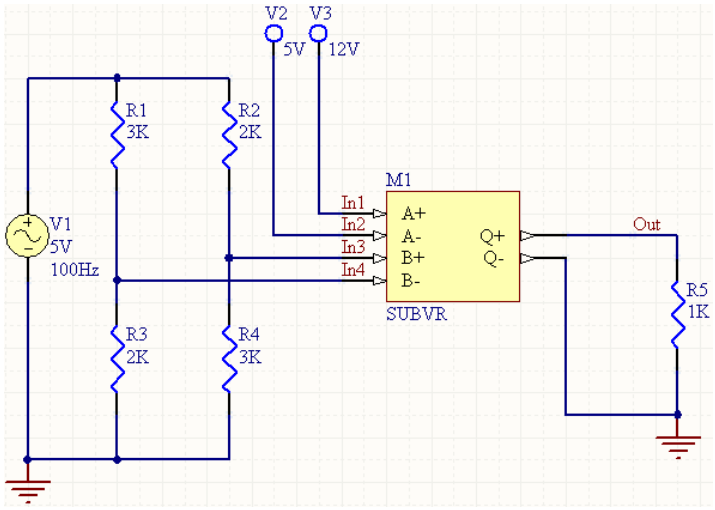
Notes

The content of the sub-circuit file (`SUBVR.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Subrtact Voltages
.SUBCKT SUBVR 1 2 3 4 5 6
```

```
BX 5 6 V=V(1,2)-V(3,4)
.ENDS SUBVR
```

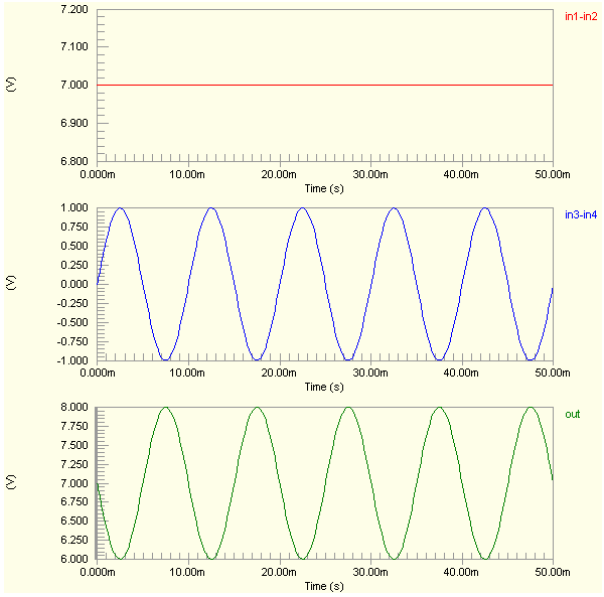
Examples



Consider the circuit in the image above. With respect to the SUBVR component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN1 IN2 IN3 IN4 OUT 0 SUBVR
.
.
*Models and Subcircuit:
.SUBCKT SUBVR 1 2 3 4 5 6
BX 5 6 V=V(1,2)-V(3,4)
.ENDS SUBVR
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



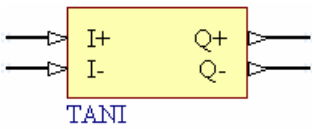
In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`

- **Transient Max Step Time** – set to `200.0u`.

Tangent

Tangent of Current



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

TANI

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

None

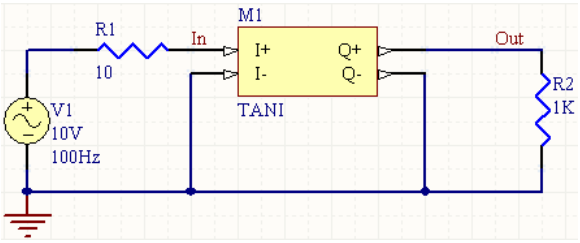
Notes

The content of the sub-circuit file (`TANI.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Tangent of Current
.SUBCKT TANI 1 2 3 4
VX 1 2 0
BX 4 3 I=TAN(I(VX))
.ENDS TANI
```

The resulting current is the value expressed in radians.

Examples

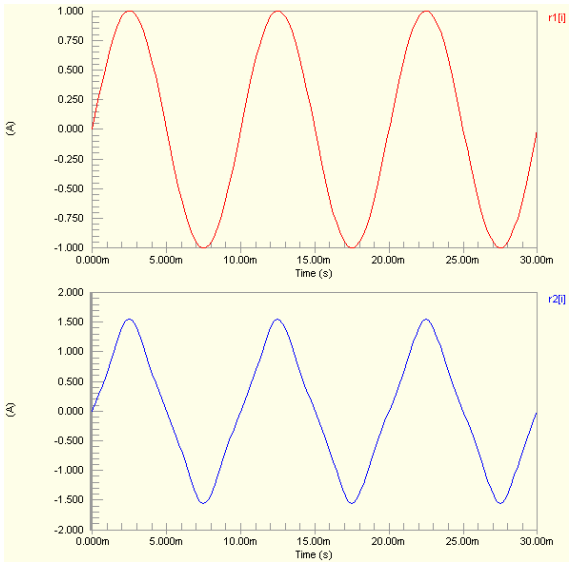


Consider the circuit in the image above. With respect to the TANI component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN 0 OUT 0 TANI
.
.
*Models and Subcircuit:
.SUBCKT TANI 1 2 3 4
VX 1 2 0
```

```
BX 4 3 I=TAN(I(VX))
.ENDS TANI
```

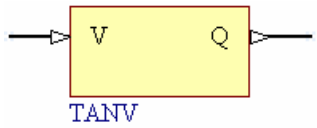
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit. .



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Tangent of Voltage (Single-Ended Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

TANV

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 @MODEL
```

Parameters (definable at component level)

None

Notes

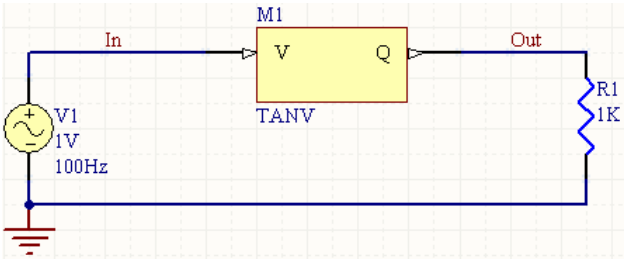
The content of the sub-circuit file (`TANV.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Tangent of Voltage
```

```
.SUBCKT TANV 1 2
BX 2 0 V=TAN(V(1))
.ENDS TANV
```

The resulting voltage is the value expressed in radians.

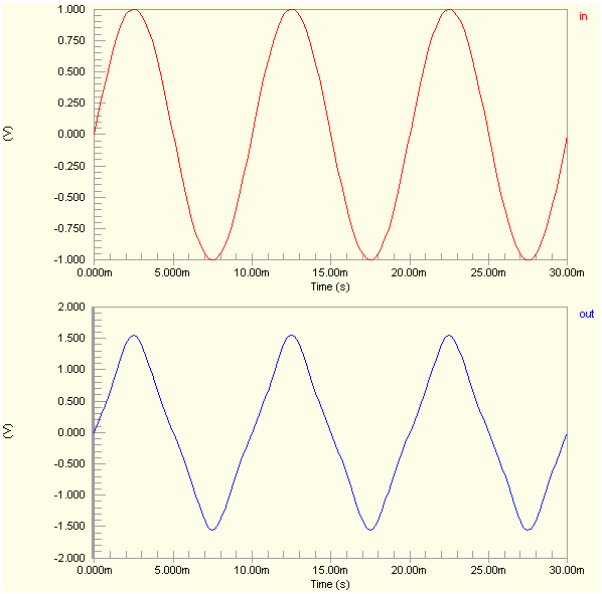
Examples



Consider the circuit in the image above. With respect to the TANV component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN OUT TANV
.
.
*Models and Subcircuit:
.SUBCKT TANV 1 2
BX 2 0 V=TAN(V(1))
.ENDS TANV
```

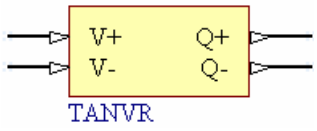
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit. .



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Tangent of Voltage (Differential Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

TANVR

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

None

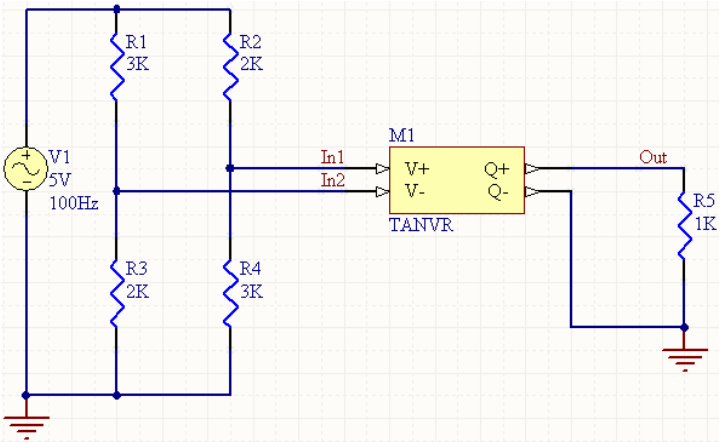
Notes

The content of the sub-circuit file (`TANVR.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Tangent of Voltage
.SUBCKT TANVR 1 2 3 4
BX 3 4 V=TAN(V(1,2))
.ENDS TANVR
```

The resulting voltage is the value expressed in radians.

Examples

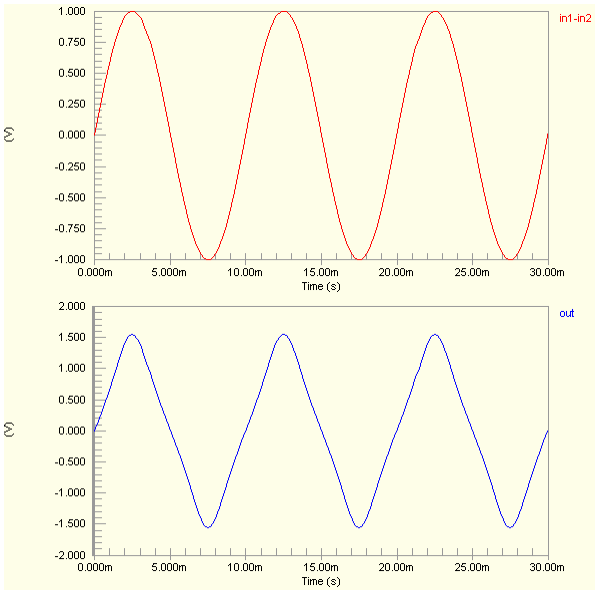


Consider the circuit in the image above. With respect to the TANVR component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN1 IN2 OUT 0 TANVR
.
.
*Models and Subcircuit:
.SUBCKT TANVR 1 2 3 4
```

```
BX 3 4 V=TAN(V(1,2))
.ENDS TANVR
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit. .

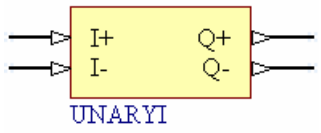


In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Unary Minus

Unary Minus of Current



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

UNARYI

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

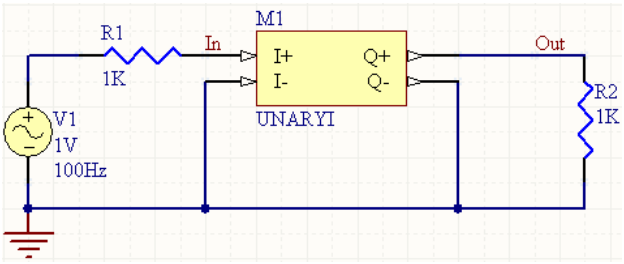
None

Notes

The content of the sub-circuit file (`UNARYI.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Unary - of Current
.SUBCKT UNARYI 1 2 3 4
VX 1 2 0
BX 4 3 I=-(I(VX))
.ENDS UNARYI
```

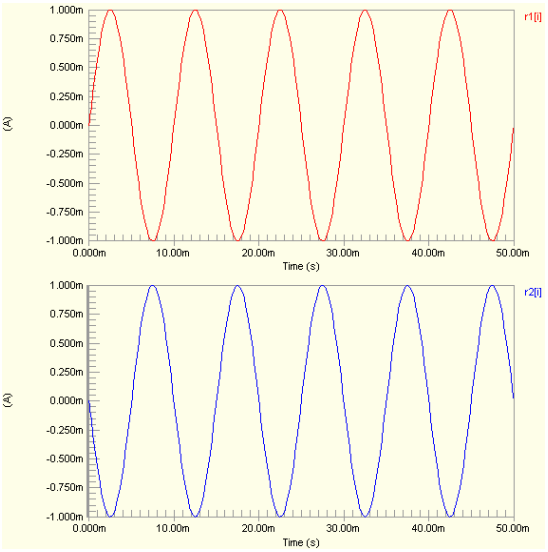
Examples



Consider the circuit in the image above. With respect to the UNARYI component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN 0 OUT 0 UNARYI
.
.
*Models and Subcircuit:
.SUBCKT UNARYI 1 2 3 4
VX 1 2 0
BX 4 3 I=-(I(VX))
.ENDS UNARYI
```

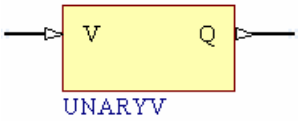
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Unary Minus of Voltage (Single-Ended Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

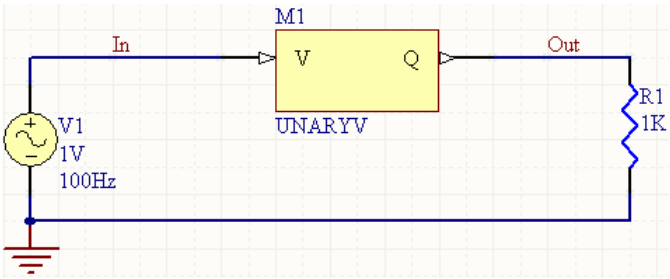
None

Notes

The content of the sub-circuit file (`UNARYV.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file:

```
*Unary Minus of Voltage
.SUBCKT UNARYV 1 2
BX 2 0 V=-(V(1))
.ENDS UNARYV
```

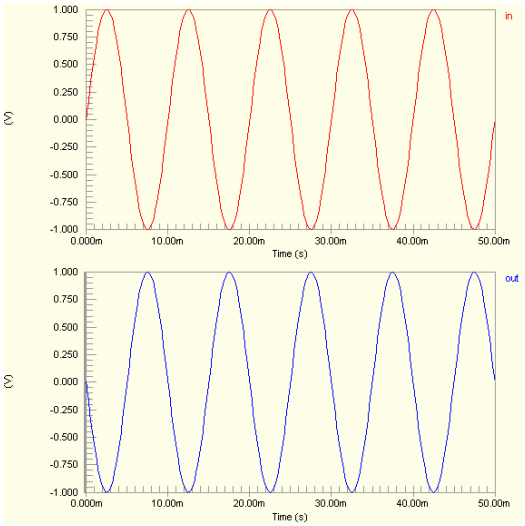
Examples



Consider the circuit in the image above. With respect to the UNARYV component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN OUT UNARYV
.
.
*Models and Subcircuit:
.SUBCKT UNARYV 1 2
BX 2 0 V=-(V(1))
.ENDS UNARYV
```

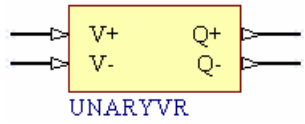
The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Unary Minus of Voltage (Differential Input)



Model Kind

General

Model Sub-Kind

Spice Subcircuit

SPICE Prefix

X

Model Name

UNARYVR

SPICE Netlist Template Format

```
@DESIGNATOR %1 %2 %3 %4 @MODEL
```

Parameters (definable at component level)

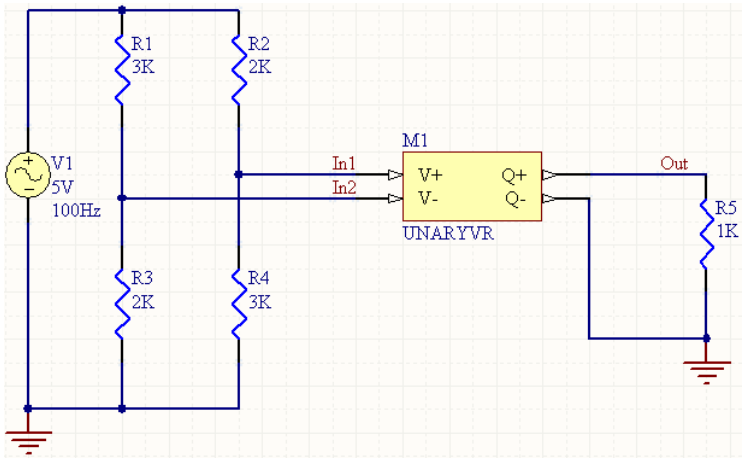
None

Notes

The content of the sub-circuit file (`UNARYVR.ckt`) associated with this model is shown below. The formula/equation used to provide this function is declared as part of the netlist-specific entry, under the `.SUBCKT` line of the file.

```
*Unary - of Voltage
.SUBCKT UNARYVR 1 2 3 4
BX 3 4 V=-(V(1,2))
.ENDS UNARYVR
```

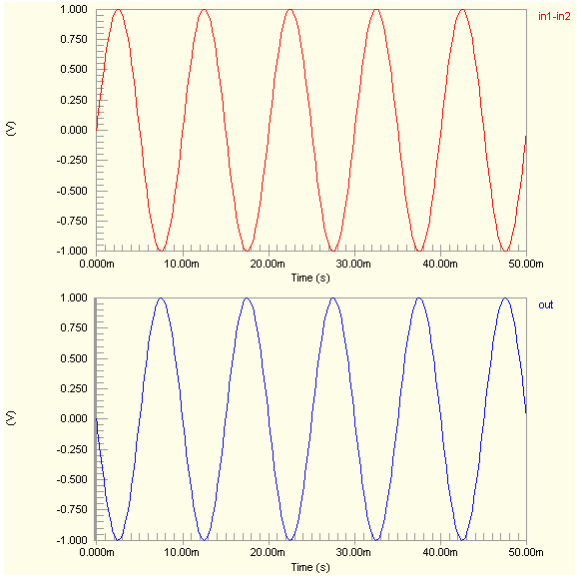
Examples



Consider the circuit in the image above. With respect to the UNARYVR component, the entries in the SPICE netlist will be:

```
*Schematic Netlist:
XM1 IN1 IN2 OUT 0 UNARYVR
.
.
*Models and Subcircuit:
.SUBCKT UNARYVR 1 2 3 4
BX 3 4 V=-(V(1,2))
.ENDS UNARYVR
```

The effect of the function can be seen in the resultant waveforms obtained by running a transient analysis of the circuit.



In this example, the following analysis parameters on the **Transient/Fourier Analysis** page of the *Analyses Setup* dialog have been used:

- **Transient Start Time** – set to `0.000`
- **Transient Stop Time** – set to `50.00m`
- **Transient Step Time** – set to `200.0u`
- **Transient Max Step Time** – set to `200.0u`.

Digital models (SimCode)

These are digital device models that have been created using the Digital SimCode™ language. This is a special descriptive language that allows digital devices to be simulated using an extended version of the event-driven XSpice. It is a form of the standard XSpice code model.

Source SimCode model definitions are stored in an ASCII text file (`*.txt`). Compiled SimCode models are stored in a compiled model file (`*.scb`). Multiple device models can be placed in the same file, with each reference by means of a special "func=" parameter.

The following generic SimCode model is considered in this section:

TTL and CMOS Logic Components

For detailed information on creating a SimCode model and linking it to a schematic component, refer to the *Creating and Linking a Digital SimCode Model* application note.

For detailed information on SimCode syntax and functions, refer to the *Digital SimCode Reference*.

Notes

The SPICE prefix for theses models is A.

All of the SimCode source and compiled model files can be found in the `Sim` folder of the installation (`\Library\Sim`). These include the main compiled model files for TTL (`LS.scb`) and CMOS (`CMOS.scb`) devices.

An intermediate simulation model file (`*.mdl`) is used to link from the schematic component to the SimCode model – effectively calling the SimCode description from within its `.MODEL` line entry.

Digital SimCode is a proprietary language - devices created with it are not compatible with other simulators, nor are digital components created for other simulators compatible with the Altium Designer-based mixed-signal Simulator.

TTL and CMOS Logic Components

Model Kind

General

Model Sub-Kind

Generic Editor

SPICE Prefix

A

SPICE Netlist Template Format

```
@DESIGNATOR [input node list][output node list] @MODEL
```

Parameters (definable at component level)

The following component-level parameters are definable for this model type and are listed on the **Parameters** tab of the *Sim Model* dialog. To access this dialog, simply double-click on the entry for the simulation model link in the **Models** region of the *Component Properties* dialog.

Propagation	-	device propagation delay. Set to `MIN` or `MAX` to use min or max data book values. (Default = typical value).
Loading	-	input-loading characteristics. Set to `MIN` or `MAX` to use min or max data book values. (Default = typical value).
Drive	-	output-drive characteristics. Set to `MIN` or `MAX` to use min or max data book values. (Default = typical value).
Current	-	device current used to specify device power. Set to `MIN` or `MAX` to use min or max data book values. (Default = typical value).
PWR value	-	power supply voltage. Specifying a value here will override any value specified by default in the model. If this value is specified, you must also specify a value for GND.

GND value	-	ground supply voltage. Specifying a value here will override any value specified by default in the model. If this value is specified, you must also specify a value for PWR.
VIL value	-	low-level input voltage. Specifying a value here will override any value specified by default in the model.
VIH value	-	high-level input voltage. Specifying a value here will override any value specified by default in the model.
VOL value	-	low-level output voltage. Specifying a value here will override any value specified by default in the model.
VOH value	-	high-level output voltage. Specifying a value here will override any value specified by default in the model.
WARN	-	set to ON to flag errors for: setup time, hold time, recovery time, pulse width, min max frequency violation and min max voltage supply violation. Errors are reported as long as the code for these conditions has been included in the SimCode model. (Default = OFF).

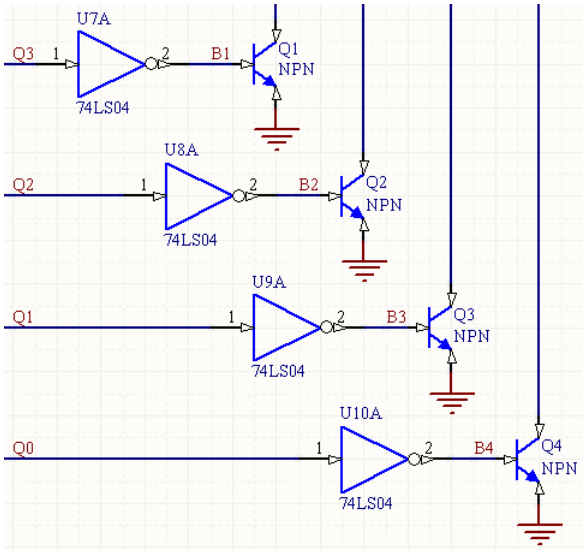
Notes

Digital devices are modeled using the Digital SimCode language. The source for a model is written using this language and stored in an ASCII text file (`*.txt`). The entry in the SPICE netlist will either point to this file or, more commonly, a file incorporating the compiled model information (`*.scb`).

All of the parameters will normally have a default value assigned. The default should be applicable to most simulations. Generally you do not need to change this value.

Entering a value for any of PWR, GND, VIL, VIH, VOL, and VOH on the **Parameters** tab of the *Sim Model* dialog, will override its specified value specified by the SimCode model, which is generally determined by the family type and supply value. For example, if a CMOS digital device was connected to a 5 volt supply, a high level on one of its outputs would, by default, be 5 volts. However, if VOH was set to `8` then a high level on one of its outputs would be 8 volts.

Examples



Consider part U9A of the DIP14 logical invertor package in the above image, with the following characteristics:

- Pin1 (Input) is connected to net `Q1`
- Pin2 (Output) is connected to net `B3`
- Pin7 of the package is connected to net `GND`
- Pin14 of the package is connected to net `VCC`
- Designator is `U9A`
- No parameter values are specified on the **Parameters** tab of the *Sim Model* dialog.
- The SPICE Netlist Template Format for this device is:

```
@DESIGNATOR [%1i %2i %3i][%1o %3o %4o] @MODEL
```

- The linked simulation model file is `74LS04.mdl`, with the following definition:

```
*LS Hex Inverter - type:digital pkg:DIP14
[DVCC=14;DGND=7;](A:1,2)(B:3,4)(C:5,6)(D:9,8)(E:11,10)(F:13,12)
.MODEL 74LS04 xsimcode(file="{MODEL_PATH}LS.SCB" func=ls04 {mntymx})
* Origin: 4049.mod.
```

The entries in the SPICE netlist would be:

```
*Schematic Netlist:
AU9A [VCC$AD GND$AD Q1$DV][VCC$DA Q1$DV B3$DA] 74LS04
.
.
*Models and Subcircuit:
.MODEL 74LS04 xsimcode(file="C:\Program Files\Altium Designer\Library\Sim\LS.SCB" func=ls04
+ )
```

The SPICE engine is directed to use the compiled model for the device, located in the file `LS.scb`, along the indicated path. The particular SimCode function to use, as there are possibly many more compiled device descriptions in this file, is also given - `ls04`. As no parameter values were entered in the *Sim Model* dialog, all parameters will be assigned their default values, as specified in the SimCode source.Simulation Analyses

The various analyses that can be performed by the Simulator, are defined in the *Analyses Setup* dialog. This dialog, through its *General Setup* page, also allows you to specify the scope of the simulation and the signals to be automatically displayed upon completion of the simulation.

Each individual analysis type is configured on a separate page of the dialog. Simply click on the analysis name to activate the corresponding setup page. The following basic analysis types are supported:

- *Operating Point Analysis*
- *Transient Analysis*
- *Fourier Analysis*
- *DC Sweep Analysis*
- *AC Small Signal Analysis*
- *Impedance Plot Analysis*
- *Noise Analysis*
- *Pole-Zero Analysis*
- *Transfer Function Analysis*

In addition, the following, more advanced, features are available:

- *Monte Carlo Analysis*
- *Parameter Sweep*
- *Temperature Sweep*.

The *SPICE Options* page of the dialog enables you to define advanced simulation options, including the values of SPICE variables, the integration method used by the Simulation Engine and the simulation reference net. In general, you should not have to change any of the parameters in this page of the dialog for accurate simulation. Only change these options if you understand SPICE simulation parameters.

Notes

The setup options that you define in the *Analyses Setup* dialog will be used in the creation of a SPICE netlist (`*.nsx`), upon which the simulation is run. In order for a SPICE netlist to be created, the schematic design must be simulatable. If there are any errors or warnings that exist, the *Analyses Setup* dialog will not appear and instead, a dialog will appear alerting you to the fact that there were errors parsing the circuit. The errors/warnings will be listed in the **Messages** panel. In this case, you will have to work through all warnings and errors and fix them, before you are able to access the *Analyses Setup* dialog and subsequently perform a simulation.

The generated SPICE netlist incorporates analysis setup information. This information is initially sought in the project file. If the design is being simulated for the first time and you have not run the *Analyses Setup* dialog, then default analysis information will be used (Transient and Operating Point analyses). After this initial simulation, and whenever you change the setup information

in the *Analyses Setup* dialog, the project will appear as being modified. Saving the project will result in the information being stored in the project file. Subsequent simulation of the design will generate the netlist using this stored information.

If you use the *Analyses Setup* dialog and then run a simulation without having saved the project, it is the setup information last defined in the dialog, and not that existing in the project, that is used.

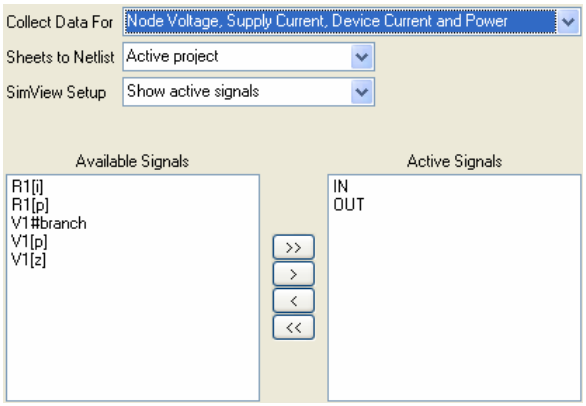
Running a simulation from the schematic will use the schematic-generated SPICE netlist, regardless of whether another SPICE netlist file is open in the main design window. The netlist will be regenerated each time a simulation is run. Any warnings or errors - either with respect to generation of the SPICE netlist, or the actual simulation process itself - will be displayed in the **Messages** panel.

A simulation can be run directly from an open SPICE netlist, regardless of whether it is part of the project or a free document. The `.nsx` file can be edited manually, prior to running a simulation from it, but care should be taken and indeed you should have good knowledge of SPICE in order to proceed down this path. If you do make modifications to the netlist and then close it, you should save it under a different name, otherwise running a simulation from the schematic will result in the modified file being overwritten when the SPICE netlist is regenerated from the schematic. Again, any warnings or errors - with respect to the actual simulation process itself - will be displayed in the **Messages** panel.

As the simulation proceeds and the defined and enabled analyses are performed, a simulation waveform file (`*.sdf`) will open as a separate tab in the main design window, to display the results of the analyses in the Sim Data Editor's Waveform Analysis window.

General Analysis Setup Options

General setup options with respect to running a circuit simulation are defined on the **General Setup** page of the *Analyses Setup* dialog. This is the default page whenever the dialog is launched. To get back to this page from the setup page of another analysis type, simply click the **General Setup** entry in the **Analyses/Options** list.



Use the **Sheets to Netlist** field to specify which schematic sheets should be included in the SPICE netlist that is passed to the Simulator. You can choose to run a simulation on the active schematic sheet or the entire set of source schematics in the active project.

Specifying simulation data to be collected

Because an enormous amount of data can be collected during a simulation, you can specify which points on the circuit and what type of data you wish to save as simulation results.

The data to be saved is specified in the **Collect Data For** field. The following options are available:

- **Node Voltage and Supply Current** - saves data for the voltage at each node and the current in each supply.
- **Node Voltage, Supply and Device Current** - saves data for the voltage at each node and the current in each supply and each device.
- **Node Voltage, Supply Current, Device Current and Power** - saves data for the voltage at each node, the current in each supply and the current and power in each device.
- **Node Voltage, Supply Current and Subcircuit VARs** - saves data for the voltage at each node, the current sourced from each supply and the voltages/currents calculated in subcircuit variables.
- **Active Signals** - saves results ONLY for signals shown in the Active Signals list. Use this option when you want to minimize the size of the result file. Signals are restricted to node voltages and supply currents.

Specifying simulation data to be displayed

When setting up a simulation you can choose which variables are automatically displayed in the Sim Data Editor's Waveform Analysis window, after the analyses have been done.

The **Available Signals** region of the page shows a list of all available circuit signals that can be plotted. Which signals are available is determined by the type of data that is being collected and saved in the result document, set in the **Collect Data For** field.

To have a signal automatically plotted in the Waveform Analysis window, select the signal in the **Available Signals** list and click the **>** button to move the signal into the **Active Signals** list.

Double-clicking on a signal moves it from one list to the other. You can select multiple signals in a list by clicking-and-dragging the mouse over the signal list, or using the **Shift** and **Ctrl** keys while clicking on signals.

While including a signal in the **Active Signals** list causes the simulation results for that signal to be automatically displayed in the Waveform Analysis window, once the simulation has finished, you can use the controls in the Sim Data Editor to display any signal for which data was collected.

Available Signals – Definitions

The following is a list of the different signal types that can appear in the **Available Signals** list:

<designator>#branch - branch current through a voltage source.

<designator>[v] - current source voltage drop.

<designator>[z] - device impedance.

<designator>[i] - device current.

<designator>[p] - device power.

<designator>[id] - Diode current, or FET drain current.

<designator>[ig] - FET gate current.

<designator>[is] - FET source current.

<designator>[ib] - BJT base current.

<designator>[ic] - BJT collector current.

<designator>[ie] - BJT emitter current.

Notes

By default, the Simulator uses the Waveform Analysis window setup information from the previous simulation run to display the simulation results. If you change the **Active Signals** list from a previous simulation run, you must set the **SimView Setup** option to `Show active signals` for any changes to the displayed waveforms to take effect. When this option is on, the Waveform Analysis window is reset to its default condition and the plot waveforms are read from the dialog list, rather than from the previous simulation run.

Operating Point Analysis

Description

An Operating Point analysis is used to determine the dc operating point of a circuit, with inductors shorted and capacitors opened.

Setup

There are no parameters to define for this type of analysis. It can only be enabled or disabled, from the **Analyses/Options** list of the *Analyses Setup* dialog.

Notes

A (DC) Operating Point analysis is automatically performed prior to a Transient analysis to determine the transient initial conditions. The exception to this is when the `Use Initial Conditions` parameter is enabled on the **Transient/Fourier Analysis Setup** page of the *Analyses Setup* dialog. In this case, analysis starts from the defined initial conditions (current and voltage) of the circuit.

An Operating Point analysis is automatically performed prior to an AC Small Signal, Noise and Pole-Zero analysis, in order to determine the linearized, small signal models for all non-linear devices in the circuit. It does not take into account the existence of any AC sources.

The simulation results are displayed on the **Operating Point** tab of the Waveform Analysis window.

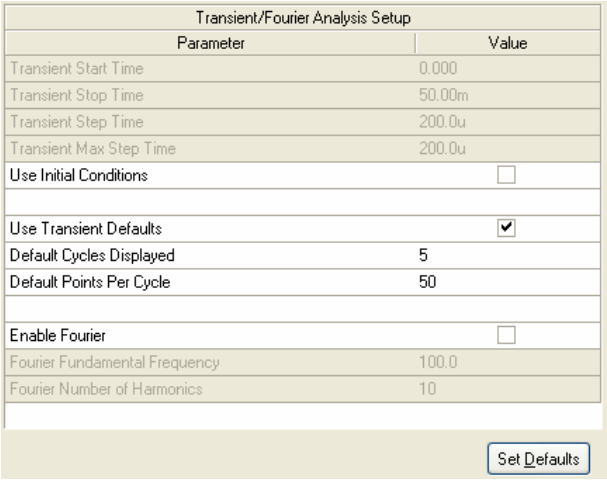
Transient Analysis

Description

A Transient analysis generates output similar to that normally shown on an oscilloscope, computing the transient output variables (voltage or current) as a function of time, over the user-specified time interval. An Operating Point analysis is automatically performed prior to a Transient analysis to determine the DC bias of the circuit, unless the `Use Initial Conditions` parameter is enabled.

Setup

Transient analysis is set up on the **Transient/Fourier Analysis Setup** page of the *Analyses Setup* dialog (after the dialog appears, simply click the **Transient/Fourier Analysis** entry in the **Analyses/Options** list). An example setup for this analysis type is shown in the following image:



Parameters

- **Transient Stat Time** - the value for the start of the required time interval for analysis (in seconds)
- **Transient Stop Time** - the value for the end of the required time interval for analysis (in seconds)
- **Transient Step Time** - the nominal time increment used in the analysis.
- **Transient Max Step Time** - the maximum variation in size of the time step that can be used by the Simulator when calculating the transient data. By default, the value used is either

  `Transient Step Time` or

  (`Transient Stop Time` - `Transient Start Time`) / 50,

  whichever is the smaller.
- **Use Initial Conditions** - when enabled, the Transient analysis begins from the initial conditions defined in the schematic, bypassing the Operating Point analysis. Use this option when you wish to perform a transient analysis starting from other than the quiescent operating point.
- **Use Transient Defaults** - when enabled, parameters are automatically calculated before each simulation run, overriding any manually set values.
- **Default Cycles Displayed** - the default number of periods of a sinusoidal waveform to display. This value is used in the automatic calculation of the `Transient Stop Time`, when the Set Defaults button is pressed.
- **Default Points Per Cycle** - the number of data points per sinusoidal waveform period. This value is used in the automatic calculation of the `Transient Step Time` when the Set Defaults button is pressed.

Notes

A Transient analysis always begins at time zero. In the time interval between zero and `Transient Start Time`, the circuit is analyzed but the results are not stored. In the time interval between `Transient Start Time` and `Transient Stop Time`, results are stored for display.

Although `Transient Step Time` is the nominal time increment used in the analysis, the actual time step is varied automatically to achieve convergence.

Typically `Transient Step Time` and `Transient Max Step Time` are set to the same value. As a starting point set both of these parameters to

(`Transient Stop Time` - `Transient Start Time`) / 1000.

If you are not sure what values to enter, press the **Set Defaults** button on the page to automatically calculate the Transient analysis parameters as follows:

- `Transient Start Time` is set to zero

- `Transient Stop Time`, `Transient Step Time` and `Transient Max Step Time` are calculated based on the values entered for the `Default Cycles Displayed` and `Default Points Per Cycle` parameters, as well as the lowest frequency source in the circuit (with frequency F_L). The formulae used for the calculations are as follows:

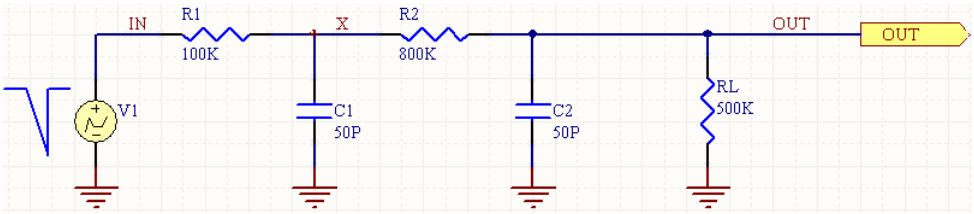
  ```
  Transient Stop Time = (1/FL) * Default Cycles Displayed
  Transient Step Time = (1/FL) / Default Points Per Cycle
  Transient Max Step Time = Transient Step Time
  ```

When using Initial Conditions, make sure that you first define the initial condition for each appropriate component in the circuit, or place .IC devices on the circuit. The IC value of a component overrides an .IC object attached to a net.

Data is saved for all signals in the **Available Signals** list, on the **General Setup** page of the *Analyses Setup* dialog.

The simulation results are displayed on the **Transient Analysis** tab of the Waveform Analysis window.

Examples



Consider the circuit in the image above, where a Transient analysis is defined with the following parameter values:

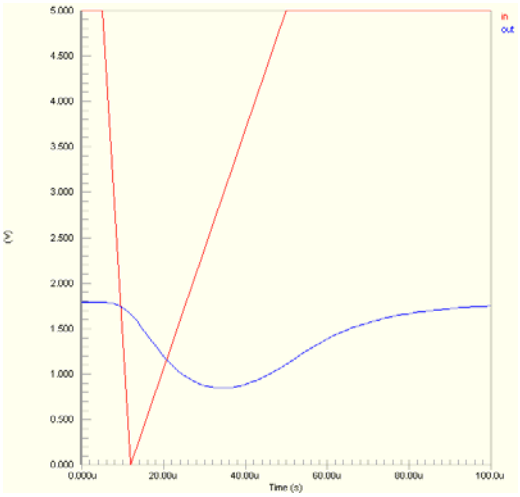
- **Transient Start Time** = `0.000`
- **Transient Stop Time** = `100.0u`
- **Transient Step Time** = `500.0n`
- **Transient Max Step Time** = `1.000u`
- **Default Cycles Displayed** = `5`
- **Default Points Per Cycle** = `50`
- **Use Initial Conditions** and **Use Transient Defaults** parameters are both disabled.
- The **Time-Value Pairs** for the PWL Voltage source are:

  ```
  0s 5V
  5us 5V
  12us 0V
  50us 5V
  60us 5V.
  ```

The entry in the SPICE netlist will be:

```
*Selected Circuit Analyses:
.TRAN 5E-7 0.0001 0 1E-6
```

and running the simulation will yield the output waveforms shown in the adjacent image.



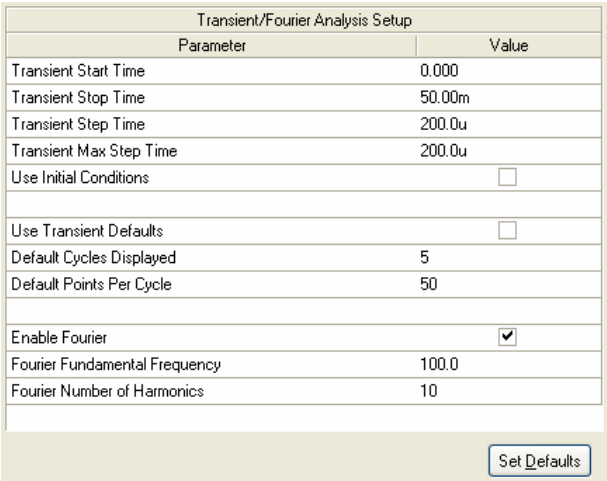
Fourier Analysis

Description

The Fourier analysis of a design is based on the last cycle of transient data captured during a Transient analysis. For example, if the fundamental frequency is 1.0kHz, then the transient data from the last 1ms cycle would be used for the Fourier analysis.

Setup

Fourier analysis is set up on the **Transient/Fourier Analysis Setup** page of the *Analyses Setup* dialog (after the dialog appears, simply click the **Transient/Fourier Analysis** entry in the **Analyses/Options** list). An example setup for this analysis type is shown in the image below:



Parameters

- **Enable Fourier** - used to include Fourier analysis in the simulation. (Default = disabled).
- **Fourier Fundamental Frequency** - the frequency of the signal that is being approximated by the sum of sinusoidal waveforms.
- **Fourier Number of Harmonics** - the number of harmonics to be considered in the analysis. Each harmonic is an integer multiple of the fundamental frequency. Together with the fundamental frequency sinusoid, the harmonics sum to form the real waveform of the signal being analyzed. The more harmonics involved in the sum, the greater the approximation to the signal's waveform (e.g. summing sinusoids to form a square wave).

Notes

If the Set Defaults button on the page is pressed, the Fourier-based parameters will be calculated as follows:

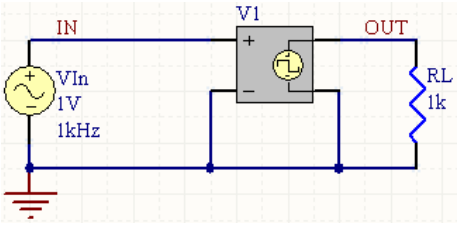
```
Fundamental Frequency = 1/(Transient Stop Time/Default Cycles Displayed)
Number of Harmonics = 10
```

Upon running the simulation, a file will be generated (`ProjectName.sim`), written to the output folder for the project and opened as the active document in the main design window. This file contains detailed information on the magnitude and phase of each harmonic in the Fourier analysis, for each of the signals in the **Available Signals** list (on the **General Setup** page of the *Analyses Setup* dialog).

You must enable the **Transient/Fourier Analysis** option in the **Analyses/Options** list of the *Analyses Setup* dialog, in order to perform a Fourier analysis.

The simulation results are displayed on the **Fourier Analysis** tab of the Waveform Analysis window.

Examples



Consider the circuit in the image above, where a Transient analysis is defined with the following parameter values:

- **Transient Start Time** = `0.000`
- **Transient Stop Time** = `5.000m`
- **Transient Step Time** = `20.00u`
- **Transient Max Step Time** = `20.00u`
- **Default Cycles Displayed** = `5`
- **Default Points Per Cycle** = `50`
- **Use Initial Conditions** and **Use Transient Defaults** parameters are both disabled.

and a Fourier analysis is enabled and defined with the parameter values:

- **Fourier Fundamental Frequency** = `1.000k`
- **Fourier Number of Harmonics** = `10`.

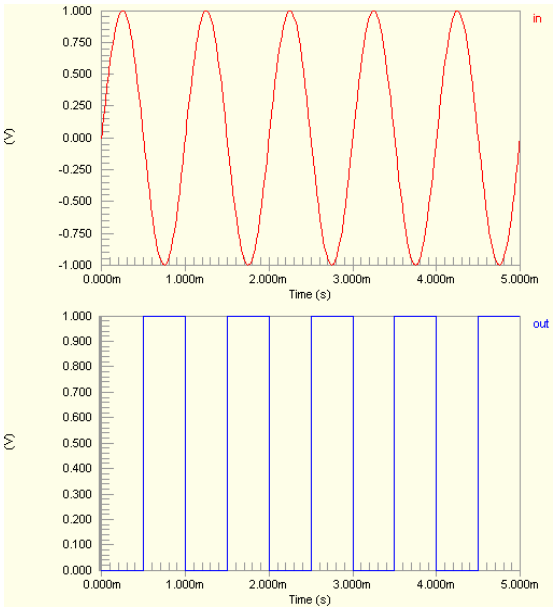
The entry in the SPICE netlist will be:

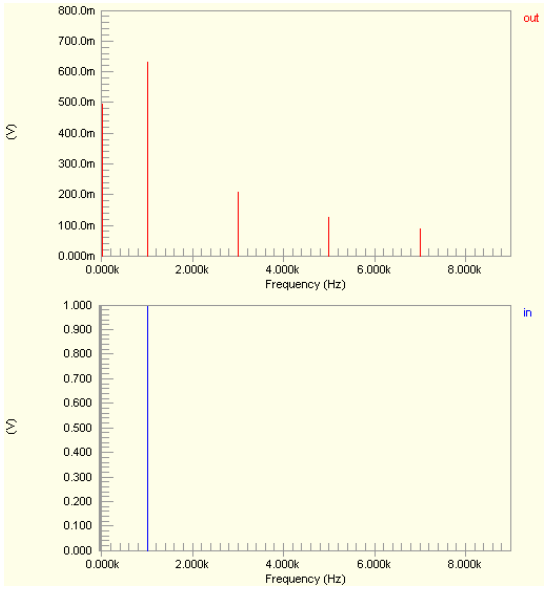
```
*Selected Circuit Analyses:
.TRAN 2E-5 0.005 0 2E-5
.SET NFREQS=10
.FOUR 1000 @VIn[p] VIn#branch @RL[p] @RL[i] OUT IN
```

The following parameter values are set for the Square Voltage Controlled Oscillator:

- **Low** = `0V`
- **High** = `1V`
- **F1** = `1KHz`

The following images show the results of the simulation. The first two plots (adjacent) show waveforms from the Transient analysis of the circuit, while the subsequent plots (below) show the results of Fourier analysis. The square wave, whose fundamental frequency is 1kHz, is broken down into sinusoids with frequencies that are odd multiples of this frequency (odd harmonics), as shown in the third plot (1kHz, 3kHz, 5kHz, 7kHz, etc) and with amplitudes that decrease with each subsequent harmonic.





DC Sweep Analysis

Description

The DC Sweep analysis generates output like that of a curve tracer. It performs a series of Operating Point analyses, modifying the voltage of a selected source in pre-defined steps, to give a DC transfer curve. You can also specify an optional secondary source.

Setup

DC Sweep analysis is set up on the **DC Sweep Analysis Setup** page of the *Analyses Setup* dialog (after the dialog appears, simply click the **DC Sweep Analysis** entry in the **Analyses/Options** list). An example setup for this analysis type is shown in the image below:

DC Sweep Analysis Setup	
Parameter	Value
Primary Source	VIN
Primary Start	-700.0m
Primary Stop	-1.500
Primary Step	-20.00m
Enable Secondary	☐
Secondary Name	
Secondary Start	0.000
Secondary Stop	0.000
Secondary Step	0.000

Parameters

- **Primary Source** - the name of the independent power source in the circuit that is to be stepped.
- **Primary Start** - the starting value for the primary power source.
- **Primary Stop** - the final value for the primary power source.
- **Primary Step** - specifies the incremental value to use over the defined sweep range.
- **Enable Secondary** - allows you to sweep the primary power source over its full range of values, for each value of a specified secondary source.
- **Secondary Name** - the name of a second independent power source in the circuit.
- **Secondary Start** - the starting value for the secondary power source.
- **Secondary Stop** - the final value for the secondary power source.
- **Secondary Step** - specifies the incremental value to use over the defined sweep range.

Notes

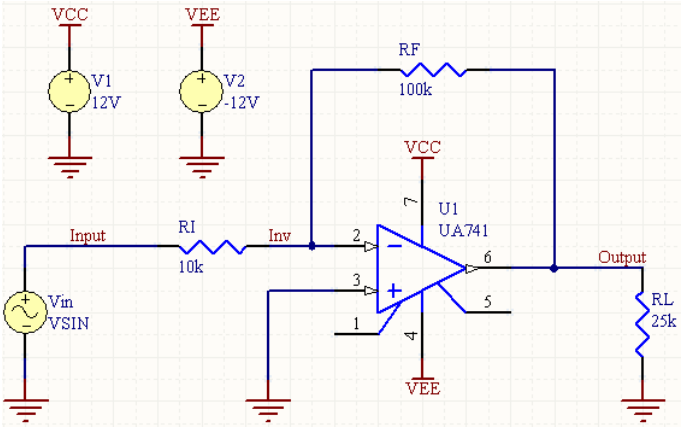
The primary source is required and the secondary source is optional.

The `Primary Source` and `Secondary Name` parameters are chosen from drop-down lists containing all power and excitation sources in the circuit.

Data is saved for all signals in the **Available Signals** list, on the **General Setup** page of the *Analyses Setup* dialog.

The simulation results are displayed on the **DC Sweep** tab of the Waveform Analysis window.

Examples



Consider the circuit in the image above, where a DC Sweep analysis is defined with the following parameter values:

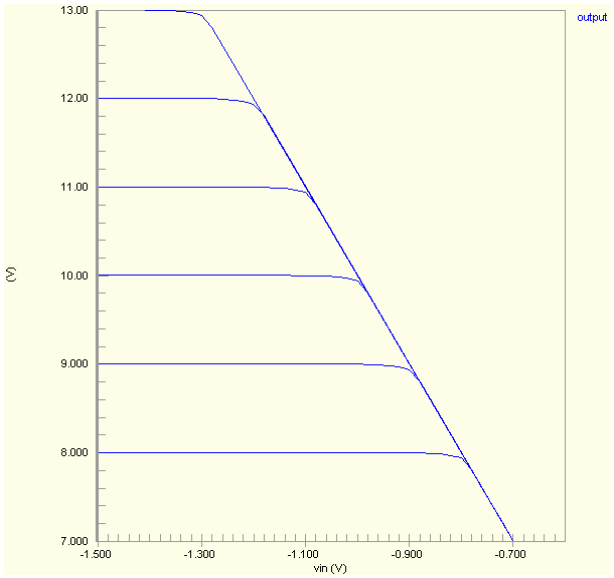
- **Primary Source** = `Vin`
- **Primary Start** = `-700.0m`
- **Primary Stop** = `-1.500`
- **Primary Step** = `-20.00m`

- **Secondary Name** = `V1`
- **Secondary Start** = `10.00`
- **Secondary Stop** = `15.00`
- **Secondary Step** = `1.000`.

The entry in the SPICE netlist will be:

```
*Selected Circuit Analyses:
.DC VIN -0.7 -1.5 -0.02 V1 10 15 1
```

and running the simulation will yield the output waveform shown in the adjacent image.



AC Small Signal Analysis

Description

An AC Small Signal analysis generates output that shows the frequency response of the circuit, calculating the small-signal AC output variables as a function of frequency.

It first performs an Operating Point analysis to determine the DC bias of the circuit, replaces the signal source with a fixed amplitude sine wave generator, then analyzes the circuit over the specified frequency range. The desired output of an AC Small Signal analysis is usually a transfer function (voltage gain, transimpedance, etc).

Setup

AC Small Signal analysis is set up on the **AC Small Signal Analysis Setup** page of the *Analyses Setup* dialog (after the dialog appears, simply click the **AC Small Signal Analysis** entry in the **Analyses/Options** list). An example setup for this analysis type is shown in the image below:

AC Small Signal Analysis Setup	
Parameter	Value
Start Frequency	100.0m
Stop Frequency	1.000meg
Sweep Type	Decade
Test Points	100
Total Test Points	701

Parameters

- **Start Frequency** - the initial frequency for the sine wave generator (in Hz).
- **Stop Frequency** - the final frequency for the sine wave generator (in Hz).
- **Sweep Type** - defines how the total number of test points is determined from the initial value assigned to the `Test Points` parameter. The following three types are available:

  `Linear` - Total number of test points evenly spaced on a linear scale.

`Decade` - Number of evenly spaced test points per decade of a $\log_{10}$ scale.

`Octave` - Number of evenly spaced test points per octave of a $\log_2$ scale.

- **Test Points** - defines the incremental value for the sweep range, in conjunction with the chosen Sweep Type.
- **Total Test Points** (non-editable) - shows the total number of test points in the frequency sweep range, calculated from the initial value for `Test Points` and the chosen `Sweep Type`.

Notes

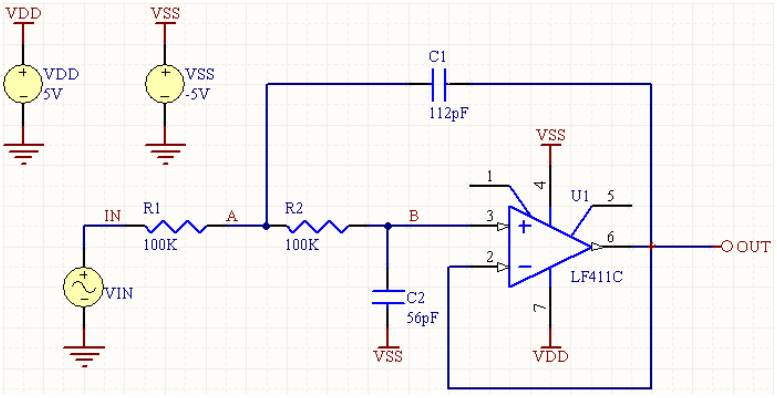
Before you can perform an AC Small Signal analysis, the circuit schematic must contain at least one signal source component with a value entered for the `AC Magnitude` parameter of its linked simulation model. It is this source that is replaced with a sine wave generator during the simulation.

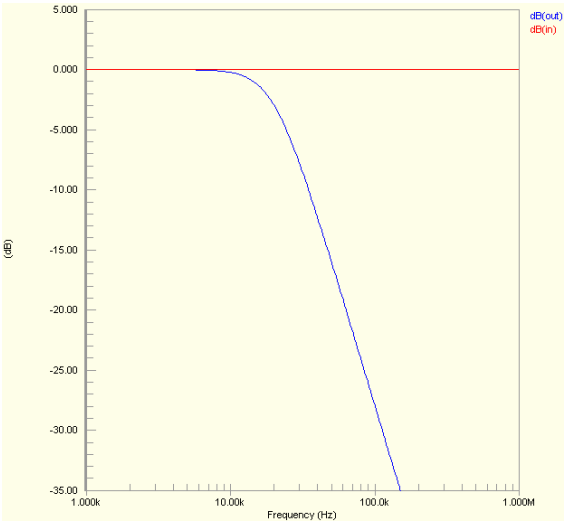
The amplitude and phase of the swept sine wave are specified in the model parameters for the SIM model linked to the schematic component for the Source. To set these values, double-click on the source component in the schematic, to bring up the *Component Properties* dialog. In the **Models** region of the dialog, double-click on the entry for the associated simulation model to launch the *Sim Model* dialog. When this dialog appears, select the **Parameters** tab to gain access to the `AC Magnitude` and `AC Phase` parameters. Enter the amplitude (in Volts) and the phase in (in Degrees). Units are not required. Set the `AC Magnitude` to `1` to have the output variables displayed relative to 0 dB.

Data is saved for all signals in the **Available Signals** list, on the **General Setup** page of the *Analyses Setup* dialog.

The simulation results are displayed on the **AC Analysis** tab of the Waveform Analysis window.

Examples





Consider the circuit in the image above, where an AC Small Signal analysis is defined with the following parameter values:

- **Start Frequency** = `1.000`
- **Stop Frequency** = `1.000meg`
- **Sweep Type** = `Decade`
- **Test Points** = `100`
- **Total Test Points** = `601`.

The entry in the SPICE netlist will be:

```
*Selected Circuit Analyses:
.AC DEC 100 1 1E6
```

and running the simulation will yield the output waveforms shown in the adjacent image.

Impedance Plot Analysis

Description

An Impedance Plot analysis shows the impedance seen by any two-terminal source in the circuit.

Setup

An Impedance Plot does not have a separate setup page of its own and is normally run and plotted as part of an AC Small Signal analysis.

To include Impedance Plot analysis results in an AC Small Signal analysis, ensure that the **Collect Data For** field on the **General Setup** page of the *Analyses Setup* dialog, is set to one of the following:

- `Node Voltage, Supply and Device Current`
- `Node Voltage, Supply Current, Device Current and Power`
- `Node Voltage, Supply Current and Subcircuit VARs`
- `Active Signals`

Locate the source of interest in the **Available Signals** list and add it to the **Active Signals** list. The signal will appear with a `[z]` suffix, indicating that it is an impedance-based signal. For example, a source in the circuit with the designator VIn, would appear in the **Available Signals** list as `VIn[z]`.

Notes

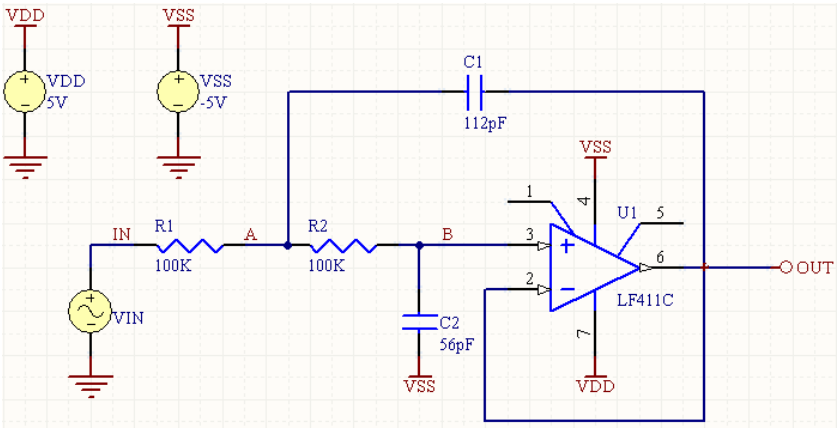
The impedance measurement is calculated from the voltage at the supply's positive terminal, divided by the current out of that same terminal.

To obtain an impedance plot of the circuit's output impedance, follow these steps:

- Remove the source from the input.
- Ground the circuit's inputs where the input supply was connected.
- Remove any load connected to the circuit.
- Connect a two-terminal source to the output, with the source's positive terminal connected to the output and its negative terminal connected to ground.
- Setup the signals of interest as described previously and run the simulation.

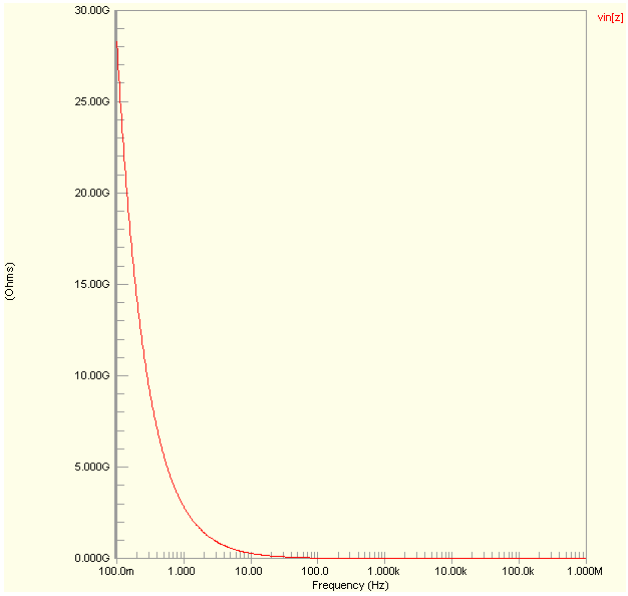
The simulation results are displayed on the **AC Analysis** tab of the Waveform Analysis window.

Examples



Consider the circuit in the above image, where an AC Small Signal analysis is to be run. To obtain an Impedance Plot analysis of the source VIN, the signal VIN[z] is taken across into the **Active Signals** list on the **General Setup** page of the *Analyses Setup* dialog.

Running the simulation yields the impedance plot shown in the following image:



Noise Analysis

Description

Noise analysis lets you measure the noise contributions of resistors and semiconductor devices by plotting the Noise Spectral Density, which is the noise measured in Volts squared per Hertz (V^2/Hz). Capacitors, inductors and controlled sources are treated as noise free.

The following noise measurements can be made:

Output Noise - the noise measured at a specified output node.

Input Noise - the amount of noise that, if injected at the input, would cause the calculated noise at the output. For example, if the output noise is 10p, and the circuit has a gain of 10, then it would take 1p of noise at the input to measure 10p of noise at the output. Thus the equivalent input noise is 1p.

Component Noise - the output noise contribution of each component in the circuit. The total output noise is the sum of individual noise contributions of resistors and semiconductor devices. Each of these components contributes a certain amount of noise, which is multiplied by the gain from that component's position to the circuit's output. Thus the same component can contribute different amounts of noise to the output, depending on its location in the circuit.

Setup

Noise analysis is set up on the **Noise Analysis Setup** page of the *Analyses Setup* dialog (after the dialog appears, simply click the **Noise Analysis** entry in the **Analyses/Options** list). An example setup for this analysis type is shown in the image below:

Noise Analysis Setup	
Parameter	Value
Noise Source	Vin
Start Frequency	1.000k
Stop Frequency	1.000meg
Sweep Type	Linear
Test Points	1000
Points Per Summary	0
Output Node	OUTPUT
Reference Node	0
Total Test Points	1000

Parameters

- **Noise Source** - an independent voltage source in the circuit which is to be used as an input reference for the noise calculations.

- **Start Frequency** - the initial frequency for the range over which to perform the noise calculations (in Hz).
- **Stop Frequency** - the final frequency for the range over which to perform the noise calculations (in Hz).
- **Sweep Type** - defines how the test points are distributed over the defined frequency range. The following three types are available:

  `Linear` - evenly spaced test points on a linear scale.

  `Decade` - evenly spaced test points per decade of a $\log_{10}$ scale.

  `Octave` - evenly spaced test points per octave of a $\log_2$ scale.
- **Test Points** - defines the number of points over the defined frequency range at which noise calculations will be performed.
- **Points Per Summary** - allows you to control which noise measurement is performed. Setting this parameter to `0` will cause input and output noise to be measured only. Setting to a `1` will measure the noise contribution of each component in the circuit.
- **Output Node** - the node in the circuit at which the total output noise is to be measured.
- **Reference Node** - the node in the circuit used as a reference for calculating the total output noise at the desired `Output Node`. By default, this parameter is set to `0` (GND). If set to any other node, the total output noise is calculated as:

  `V(Output Node) -V(Reference Node)`
- **Total Test Points** (non-editable) - shows the total number of test points in the frequency sweep range, calculated from the initial value for Test Points and the chosen Sweep Type.

Notes

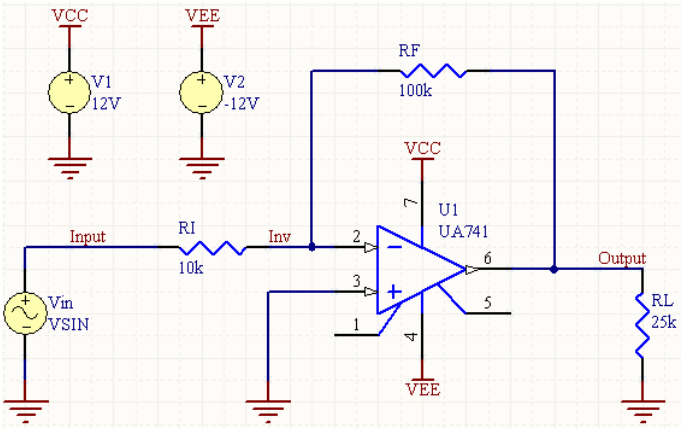
The `Start Frequency` must be greater than zero.

The independent voltage source specified in the `Noise Source` parameter must be an ac source in order for the simulation to proceed.

Data is saved for all signals in the **Available Signals** list, on the **General Setup** page of the *Analyses Setup* dialog.

The simulation results are displayed on the **Noise Spectral Density** tab of the Waveform Analysis window.

Examples



Consider the circuit in the image above, where a Noise analysis is defined with the following parameter values:

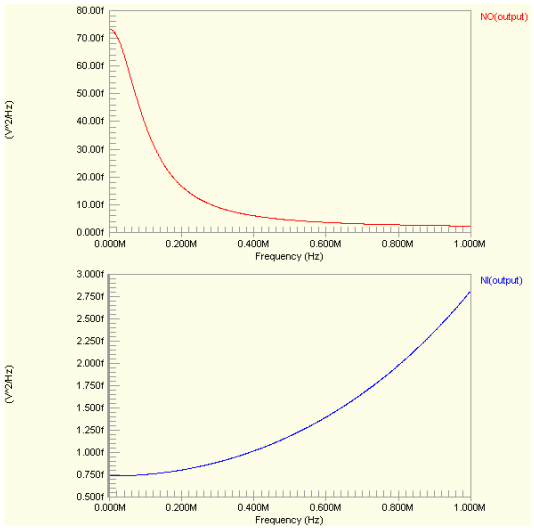
- **Noise Source** = `Vin`
- **Start Frequency** = `1.000k`
- **Stop Frequency** = `1.000meg`
- **Sweep Type** = `Linear`
- **Test Points** = `1000`
- **Points Per Summary** = `0`
- **Output Node** = `Output`
- **Reference Node** = `0` (GND)
- **Total Test Points** = `1000`.

The entry in the SPICE netlist will be:

```
*Selected Circuit Analyses:
```

```
.NOISE V(OUTPUT) Vin LIN 1000 1000 1E6
```

and running the simulation will yield the output waveforms shown in the following image.



The top waveform shows the total output noise (NO) measured at the specified output node, in this case Output. The bottom waveform shows the amount of noise that would have to be injected at the input (NI) to obtain the measured output noise at this node.

If the **Points Per Summary** parameter had been set to 1 instead of 0, the output noise contribution of each applicable component in the circuit would have been measured and the corresponding waveforms for each made available in the **Sim Data** panel, ready for use in the Waveform Analysis window.

Pole-Zero Analysis

Description

Pole-Zero analysis enables you to determine the stability of a single input, single output linear system, by calculating the poles and/or zeros in the small-signal ac transfer function for the circuit. The dc operating point of the circuit is found and then linearized, small-signal models for all non-linear devices in the circuit are determined. This circuit is then used to find the poles and zeros that satisfy the nominated transfer function.

The transfer function can either be Voltage Gain (output voltage/input voltage) or Impedance (output voltage/input current).

Setup

Pole-Zero analysis is set up on the **Pole-Zero Analysis Setup** page of the *Analyses Setup* dialog (after the dialog appears, simply click the **Pole-Zero Analysis** entry in the **Analyses/Options** list). An example setup for this analysis type is shown in the image below:

Pole-Zero Analysis Setup	
Parameter	Value
Input Node	IN
Input Reference Node	0
Output Node	OUT
Output Reference Node	0
Transfer Function Type	V(output)/V(input)
Analysis Type	Poles and Zeros

Parameters

- **Input Node** - the positive input node for the circuit.
- **Input Reference Node** - the reference node for the input of the circuit (Default = `0` (GND)).
- **Output Node** - the positive output node for the circuit.
- **Output Reference Node** - the reference node for the output of the circuit (Default = `0` (GND)).
- **Transfer Function Type** - defines the type of ac small-signal transfer function to be used for the circuit when calculating the poles and/or zeros. There are two types available:

  `V(output)/V(input)` **-** Voltage Gain Transfer Function.

`V(output)/I(input)` - Impedance Transfer Function.

- **Analysis Type** - allows you to further refine the role of the analysis. Choose to find all poles that satisfy the transfer function for the circuit (`Poles Only`), all zeros (`Zeros Only`), or both `Poles and Zeros`.

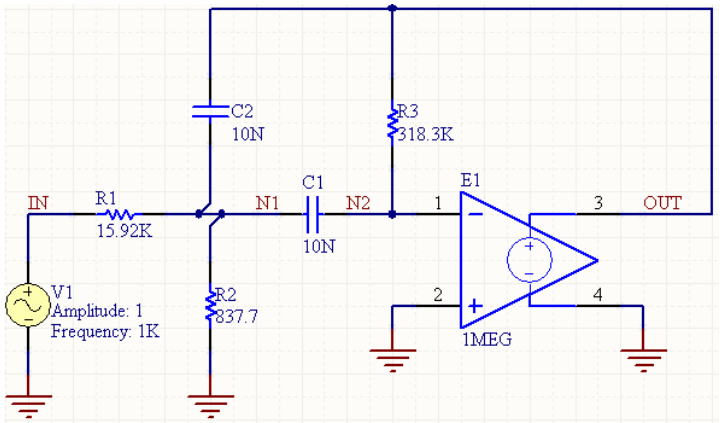
Notes

Pole-Zero analysis works with resistors, capacitors, inductors, linear-controlled sources, independent sources, diodes, BJTs, MOSFETs and JFETs. Transmission lines are not supported.

The method used in the analysis is a sub-optimal numerical search. For large circuits it may take a considerable time or fail to find all poles and zeros. For some circuits, the method becomes "lost" and finds an excessive number of poles or zeros. If there is non-convergence in finding both poles and zeros, refine the analysis to calculate only poles or only zeros.

The simulation results are displayed on the **Pole-Zero Analysis** tab of the Waveform Analysis window.

Examples



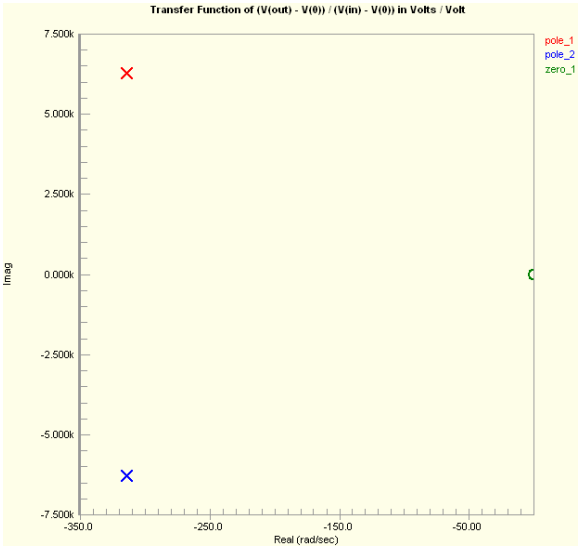
Consider the circuit in the image above, where a Pole-Zero analysis is defined with the following parameter values:

- **Input Node** = `IN`
- **Input Reference Node** = `0`
- **Output Node** = `OUT`
- **Output Reference Node** = `0`
- **Transfer Function Type** = `V(output)/V(input)`
- **Analysis Type** = `Poles and Zeros`.

The entry in the SPICE netlist will be:

```
*Selected Circuit Analyses:
.PZ IN 0 OUT 0 VOL PZ
```

and running the simulation will yield the output wave plot shown in the adjacent image.



Transfer Function Analysis

Description

The Transfer Function analysis (DC small signal analysis) calculates the DC input resistance, DC output resistance and DC gain, at each voltage node in the circuit.

Setup

Transfer Function analysis is set up on the **Transfer Function Analysis Setup** page of the *Analyses Setup* dialog (after the dialog appears, simply click the **Transfer Function Analysis** entry in the **Analyses/Options** list). An example setup for this analysis type is shown in the image below:

Transfer Function Analysis Setup	
Parameter	Value
Source Name	Vin
Reference Node	0

Parameters

- **Source Name** - the small signal input source used as the input reference for the calculations.
- **Reference Node** - the node in the circuit used as a reference for the calculations at each specified voltage node. By default, this parameter is set to 0 (GND).

Notes

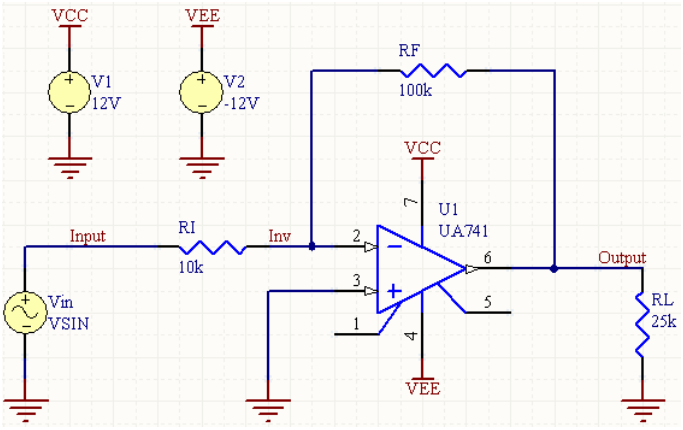
Data is saved for all voltage nodes in the circuit only.

The three small signal calculations are:

- **Gain (Transfer Function)** - the ration of the voltage at the specific Voltage Node in the circuit to the input source defined by the `Source Name` parameter.
- **Input resistance** - measured at the input source defined by the `Source Name` parameter.
- **Output resistance** - measured across the specific Voltage Node in the circuit and the node defined by the `Reference Node` parameter.

The simulation results are displayed on the **Transfer Function** tab of the Waveform Analysis window.

Examples



Consider the circuit in the image above, where a Transfer Function analysis is defined with the following parameter values:

- **Source Name** = `Vin`
- **Reference Node** = `0` (GND)

The entry in the SPICE netlist will be:

```
*Selected Circuit Analyses:
.TF V(INPUT) Vin
.TF V(INV) Vin
.TF V(OUTPUT) Vin
.TF V(VCC) Vin
.TF V(VEE) Vin
```

If the nodes `INPUT` and `OUTPUT` are taken into the **Active Signals** list on the **General Setup** page of the *Analyses Setup* dialog, then the following data will be obtained upon running the simulation:

TF_V(OUTPUT)/VIN	-9.999 : Transfer Function for V(OUTPUT)/VIN
IN(OUTPUT)_VIN	10.00k : Input resistance at VIN
OUT_V(OUTPUT)	15.38m : Output resistance at OUTPUT
TF_V(INPUT)/VIN	1.000 : Transfer Function for V(INPUT)/VIN
IN(INPUT)_VIN	10.00k : Input resistance at VIN
OUT_V(INPUT)	0.000 : Output resistance at INPUT

Monte Carlo Analysis

Description

Monte Carlo analysis allows you to perform multiple simulation runs with component values randomly varied across specified tolerances. The Simulator performs multiple passes of any of the standard analyses that are enabled (AC, DC Sweep, Operating Point, Transient, Transfer Function, Noise).

The Monte Carlo analysis can vary basic components and models - subcircuit data is not varied during the analysis.

Setup

Monte Carlo analysis is set up on the **Monte Carlo Analysis Setup** page of the *Analyses Setup* dialog (after the dialog appears, simply click the **Monte Carlo Analysis** entry in the **Analyses/Options** list). An example setup for this analysis type is shown in the image below:

Monte Carlo Analysis Setup	
Parameter	Value
Seed	-1
Distribution	Uniform
Number of Runs	5
Default Resistor Tolerance	15%
Default Capacitor Tolerance	15%
Default Inductor Tolerance	10%
Default Transistor Tolerance	10%
Default DC Source Tolerance	10%
Default Digital Tp Tolerance	10%
Specific Tolerances	0 defined...

Parameters

- **Seed** - this value is used by the Simulator to generate random numbers for the various runs of the analysis. If you want to run a simulation with a different series of random numbers, this value must be changed to another number. (Default = `-1`).
- **Distribution** - this parameter defines the distribution of values obtained during random number generation. Three distribution types are available:

  `Uniform` (Default)

  This is a flat distribution. Values are uniformly distributed over the specified tolerance range. For example, for a 1K resistor with a tolerance of 10%, there is an equal chance of the randomly generated value being anywhere between 900 Ω and 1100 Ω.

  `Gaussian`

  Values are distributed according to a Gaussian (bell-shaped) curve, with the center at the nominal value and the specified tolerance at +/- 3 standard deviations. For a resistor with a value of 1K +/- 10%, the center of the distribution would be at 1000 Ω, +3 standard deviations is 1100 Ω and -3 standard deviations is 990 Ω.

  With this type of distribution, there is a higher probability that the randomly generated value will be closer to the specified value.

  `Worst Case`

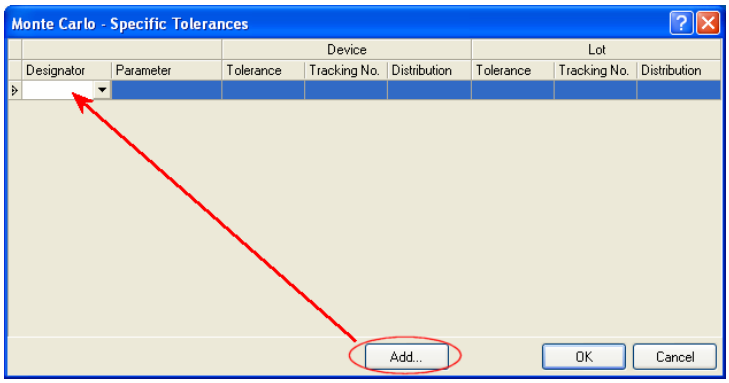
  This is the same as the Uniform distribution, but only the end points (worst case) of the range are used. For a 1K +/- 10% resistor, the value used would be randomly chosen from the two worst case values of 990 Ω and 1100 Ω. On any one simulation run there is an equal chance that the high-end worst case value (1100 Ω) or low-end worst case value (990 Ω) will be used.
- **Number of Runs** - the number of simulation runs you want the Simulator to perform. Different device values will be used for each run, within the specified tolerance range. (Default = `5`).

- **Default Resistor Tolerance** - the default tolerance to be observed for resistors. The value is entered as a percentage (Default = `10%`).
- **Default Capacitor Tolerance** - the default tolerance to be observed for capacitors. The value is entered as a percentage (Default = `10%`).
- **Default Inductor Tolerance** - the default tolerance to be observed for inductors. The value is entered as a percentage (Default = `10%`).
- **Default Transistor Tolerance** - the default tolerance to be observed for transistors (beta forward). The value is entered as a percentage (Default = `10%`).
- **Default DC Source Tolerance** - the default tolerance to be observed for DC Sources. The value is entered as a percentage (Default = `10%`).
- **Default Digital Tp Tolerance** - the default tolerance to be observed for Digital Tp (propagation delay for digital devices). The value is entered as a percentage (Default = `10%`).

  The tolerance is used to determine the allowable range of values that can be generated by the random number generator for a device. For a device with nominal value $\mathbf{Val_{Nom}}$, the range can be expressed as:

  `ValNom - (Tolerance * ValNom) ≤ RANGE ≥ ValNom + (Tolerance * ValNom)`
- **Specific Tolerances** - this parameter shows how many specific tolerances are currently defined. These are user-defined tolerances that are applied to specific components in the circuit. You can set up your own specific tolerances as required, by clicking the **...** button to the right of the field. The *Monte Carlo - Specific Tolerances* dialog will appear. Specific tolerances that are defined will override the default tolerance settings. (Default = `0 defined`).

Defining Specific Tolerances



To define a new specific tolerance, click the **Add** button at the bottom of the *Monte Carlo – Specific Tolerances* dialog. A new row will be added.

In the **Designator** field, choose the component that the specific tolerance is to apply to, from the drop-down list. Include a parameter in the **Parameter** field if the device requires it. Supported parameters include: the propagation delay of a digital component, the Beta forward of a transistor and the resistance of a potentiometer.

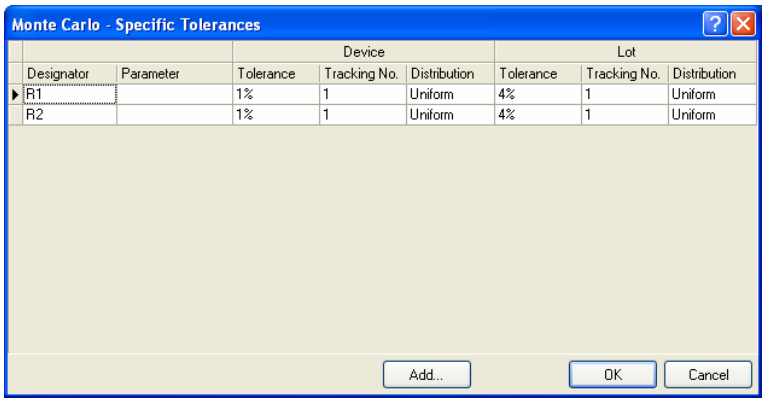
Each component can have two tolerances set: a **Device** tolerance and a **Lot** tolerance. Both device and lot tolerances are allowed, but only one is required. For a specific component, device and lot tolerances are calculated independently (using different random numbers) and then added together.

Set the **Tolerance** field to give the percentage tolerance for the component. The **Tracking No.** field is used to assign a common tracking number to components when you require the variation in their tolerance to be correlated. The **Distribution** field is used to specify the distribution type used for random number generation (`Uniform`, `Gaussian` or `Worst Case`). If you give two components the same tracking number and device distribution then the same random number is used for both components when the device values for a simulation run are calculated.

Combined device and lot tolerances are useful where values are not completely correlated, but are not completely independent either. An example would be two different resistor packs. Here, the lot tolerance can be large (that is, the variation from wafer to wafer), while the device tolerance (the variation from resistor to resistor in the same package), is small. In this case the device tolerance should not be ignored because it may limit the overall performance of a circuit.

Consider the following example:

Assume R1 and R2 are both 1K, with a Device Tolerance of 1% (same Device Tracking number) and they have a Lot Tolerance of 4%, with the same Lot Tracking number.



For each Monte Carlo run the resistors are first assigned the same lot variation (a nominal value) between +/- 4%. Then each resistor is assigned a device tolerance between +/- 1%. This gives a total tolerance of 5% (1% + 4%). However, during the same run the values of each resistor cannot be any farther than +/- 1% from their nominal value, or 2% from each other.

Notes

At least one of the standard analysis types (AC, DC Sweep, Operating Point, Transient, Transfer Function, Noise) must be enabled in order to perform a Monte Carlo analysis.

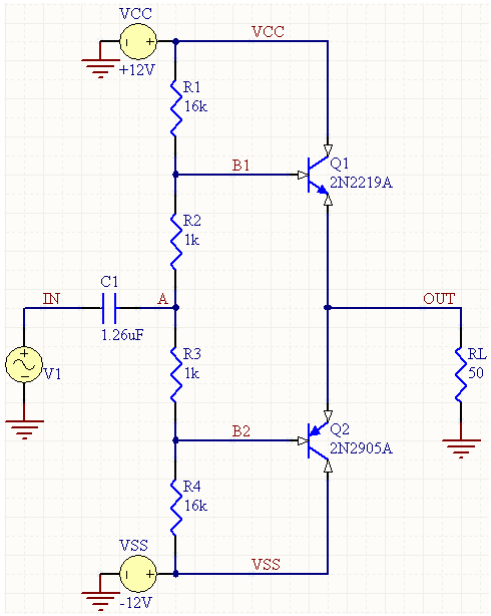
Data is saved for all signals in the **Available Signals** list, on the **General Setup** page of the *Analyses Setup* dialog.

Running a Monte Carlo analysis can result in a large amount of data being calculated. To limit the amount of data calculated, you can set the **Collect Data For** option on the **General Setup** page of the *Analyses Setup* dialog to **Active Signals**. With this option, data is only calculated for variables currently listed in the **Active Signals** list.

Each component is randomly varied independent of other components. For example, if a circuit has two 10 K resistors, and the default tolerance is set to 10%, then during the first pass of the simulation, one resistor might have a value of 953 Ω, and the other one could be 1022 Ω. The program uses a separate and independent random number to generate the value for each component.

As running a Monte Carlo analysis actually performs multiple passes of the enabled standard analyses, there is a special identifier used when displaying the waveforms in the Sim Data Editor's Waveform Analysis window. Each pass is identified by adding a letter and number as a suffix to the waveform name. For a Monte Carlo analysis, the letter used is `m` and the number used identifies which pass the waveform relates to (e.g. `Output_m1`, `Output_m2`, etc).

Examples



Consider the circuit in the image above, where a Transient analysis is to be performed in conjunction with the use of the Monte Carlo analysis feature. The Transient analysis is defined with the following parameter values:

- **Transient Start Time =** `0.000`

- **Transient Stop Time** = `500.0u`
- **Transient Step Time** = `2.000u`
- **Transient Max Step Time** = `2.000u`.

The Monte Carlo analysis is defined with the following parameter values:

- **Seed** = `-1`
- **Distribution** = `Uniform`
- **Number of Runs** = `5`
- **Default Resistor Tolerance** = `15%`
- **Default Capacitor Tolerance** = `15%`
- All other parameters are left at their default values.

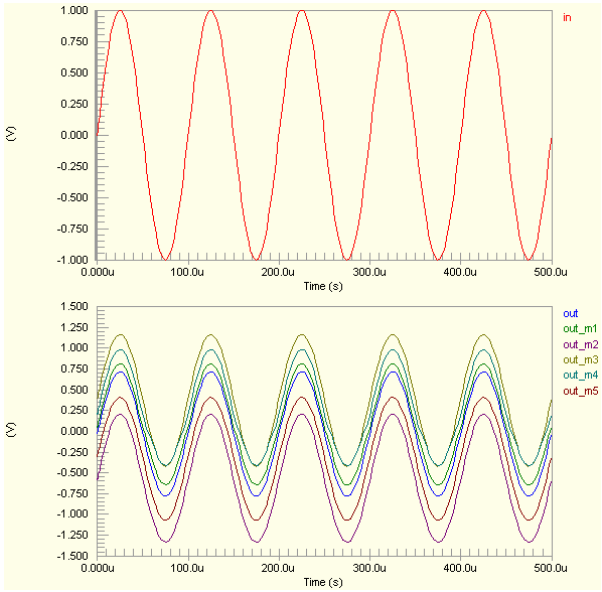
The entry in the SPICE netlist will be:

```
*Selected Circuit Analyses:
.TRAN 2E-6 0.0005 0 2E-6
.CONTROL
TOL C1 DEV=15% Uniform
TOL Q1[BF] DEV=10% Uniform
TOL Q2[BF] DEV=10% Uniform
TOL R1 DEV=15% Uniform
TOL R2 DEV=15% Uniform
TOL R3 DEV=15% Uniform
TOL R4 DEV=15% Uniform
TOL RL DEV=15% Uniform
TOL V1 DEV=10% Uniform
TOL VCC DEV=10% Uniform
TOL VSS DEV=10% Uniform
MC 5 SEED=-1
.ENDC
```

Running the simulation will yield standard waveforms for the IN and OUT signals. For the OUT signal, the following additional waveforms will be available in the **Source Data** region of the **Sim Data** panel, corresponding to the five runs performed as part of the Monte Carlo analysis.

- `out_m1`
- `out_m2`
- `out_m3`
- `out_m4`
- `out_m5`

Simply make the wave plot containing the out signal active and double-click on each additional waveform to add it to the view. The default waveform (`out`) will also be generated for comparison.



Parameter Sweep

Description

The Parameter Sweep feature allows you to sweep the value of a device in defined increments, over a specified range. The Simulator performs multiple passes of any of the standard analyses that are enabled (AC, DC Sweep, Operating Point, Transient, Transfer Function, Noise).

The Parameter Sweep can vary basic components and models - subcircuit data is not varied during the analysis. You can also define a Secondary parameter to be swept. When a Secondary parameter is defined the Primary parameter is swept for each value of the Secondary parameter.

Setup

Parameter Sweep is set up on the **Parameter Sweep Setup** page of the *Analyses Setup* dialog (after the dialog appears, simply click the **Parameter Sweep** entry in the **Analyses/Options** list). An example setup for this feature is shown in the image below:

Parameter Sweep Setup	
Parameter	Value
Primary Sweep Variable	RF[resistance]
Primary Start Value	50.00k
Primary Stop Value	150.0k
Primary Step Value	50.00k
Primary Sweep Type	Absolute Values
Enable Secondary	☑
Secondary Sweep Variable	RI[resistance]
Secondary Start Value	5.000k
Secondary Stop Value	15.00k
Secondary Step Value	5.000k
Secondary Sweep Type	Absolute Values

Parameters

- **Primary Sweep Variable** - the device or parameter in the circuit whose value you wish to have swept. All possible variables that can be swept in the circuit are automatically loaded into a convenient drop-down list, from which to choose.
- **Primary Start Value** - the initial value for the `Primary Sweep Variable`.
- **Primary Stop Value** - the final value in the required sweep range for the `Primary Sweep Variable`.
- **Primary Step Value** - the incremental step to be used in determining the sweep values across the defined sweep range.
- **Primary Sweep Type** - set to `Absolute Values` to step through the defined sweep range exactly as entered (from `Primary Start Value` to `Primary Stop Value`) and thereby obtain a set of progressive, absolute values for the parameter. Set to `Relative Values` to add the values of the sweep range to the default value of the device or parameter, thereby creating a relative set of values for the parameter. For example, consider a parameter sweep defined a s follows:

`Primary Sweep Variable` = 10k resistor

`Primary Start, Stop and Step` values are 5k, 15k an 5k respectively

With the `Primary Sweep Type` set to `Absolute Values`, the resulting resistor values would be used in the simulation passes:

```
5k, 10k, 15k
```

If `Relative Values` is chosen instead, the resulting values used would be:

```
15k, 20k, 25k
```

- **Enable Secondary** - enables the use of a secondary parameter variable in the sweep. In this case, the `Primary Sweep Variable` is swept for each value of the secondary.
- **Secondary Sweep Variable** - the device or parameter in the circuit whose value you wish to have swept and used as a control to sweeping the `Primary Sweep Variable`. All possible variables that can be swept in the circuit are automatically loaded into a convenient drop-down list, from which to choose.
- **Secondary Start Value** - the initial value for the `Secondary Sweep Variable`.
- **Secondary Stop Value** - the final value in the required sweep range for the `Secondary Sweep Variable`.
- **Secondary Step Value** - the incremental step to be used in determining the sweep values across the defined sweep range.
- **Secondary Sweep Type** - as per `Primary Sweep Type` above, but applied to the generation of values to be used for the `Secondary Sweep Variable`.

Notes

At least one of the standard analysis types (AC, DC Sweep, Operating Point, Transient, Transfer Function, Noise) must be enabled in order to perform a Parameter Sweep analysis.

The parameter to be swept can be a single designation or a designation with a device parameter in brackets. The following are some valid examples:

- `RF` - Resistor with designation RF
- `Q3[bf]` - Beta forward on transistor Q3
- `R3[r]` - Resistance of potentiometer R3
- `option[temp]` - Temperature
- `U5[tp_val]` - Propagation delays of digital device U5

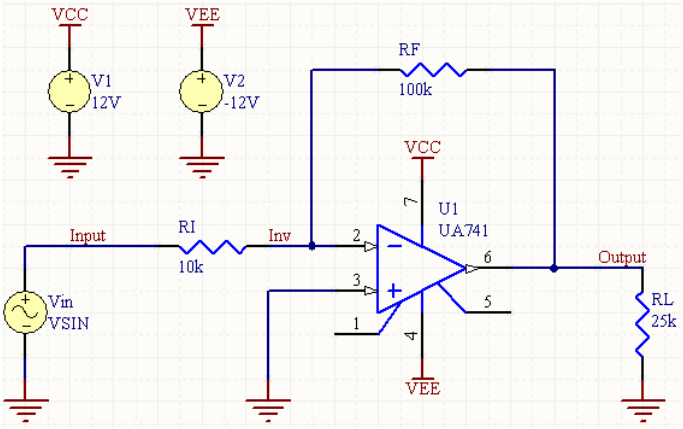
Data is saved for all signals in the **Available Signals** list, on the **General Setup** page of the *Analyses Setup* dialog.

Running a Parameter Sweep can result in a large amount of data being calculated. To limit the amount of data calculated, you can set the **Collect Data For** option on the **General Setup** page of the *Analyses Setup* dialog to **Active Signals**. With this option, data is only calculated for variables currently listed in the **Active Signals** list.

Normally you would use a Temperature Sweep to vary the temperature for simulation. However, temperature can also be varied in the Parameter Sweep. This is useful if you want to vary the temperature as either the primary or secondary parameter in a two-parameter sweep.

As running a Parameter Sweep actually performs multiple passes of the analysis (varying one or more circuit parameters with each pass), there is a special identifier used when displaying the waveforms in the Sim Data Editor's Waveform Analysis window. Each pass is identified by adding a letter and number as a suffix to the waveform name. For a Parameter Sweep, the letter used is `p` and the number used identifies which pass the waveform relates to (e.g. `Output_p1`, `Output_p2`, etc). When you click on a waveform name in the Waveform Analysis window, the values used for the parameters in that pass of the sweep are displayed – both next to the wave plot and in the Status bar.

Examples



Consider the circuit in the image above, where AC Small Signal and Transient analyses are to be performed in conjunction with the use of the Parameter Sweep feature. The Transient analysis is defined with the following parameter values:

- **Transient Start Time** = `0.000`
- **Transient Stop Time** = `500.0u`
- **Transient Step Time** = `2.000u`
- **Transient Max Step Time** = `2.000u`.

The AC Small Signal analysis is defined with the following parameter values:

- **Start Frequency** = `1.000`
- **Stop Frequency** = `1.000meg`
- **Sweep Type** = `Decade`
- **Test Points** = `100`
- **Total Test Points** = `601`.

The Parameter Sweep is defined with the following parameter values:

- **Primary Sweep Variable** = `RF[resistance]`
- **Primary Start Value** = `50.00k`
- **Primary Stop Value** = `150.0k`
- **Primary Step Value** = `50.00k`
- **Primary Sweep Type** = `Absolute Values`
- **Secondary Sweep Variable** = `RI[resistance]`
- **Secondary Start Value** = `5.000k`
- **Secondary Stop Value** = `15.00k`
- **Secondary Step Value** = `5.000k`
- **Secondary Sweep Type** = `Absolute Values`.

The entry in the SPICE netlist will be:

```
*Selected Circuit Analyses:
.AC DEC 100 1 1E6
.TRAN 2E-6 0.0005 0 2E-6
.CONTROL
SWEEP RF[resistance] 5E4 1.5E5 5E4 RI[resistance] 5000 1.5E4 5000
.ENDC
```

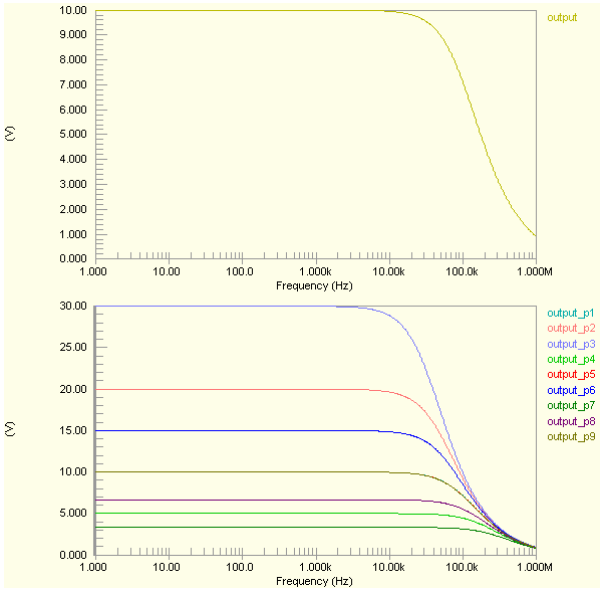
There are three values of the primary parameter that will be swept for each of the three values of the secondary. Therefore there will be nine waveforms in all generated by the sweep:

- `output_p1`
- `output_p2`
- `output_p3`

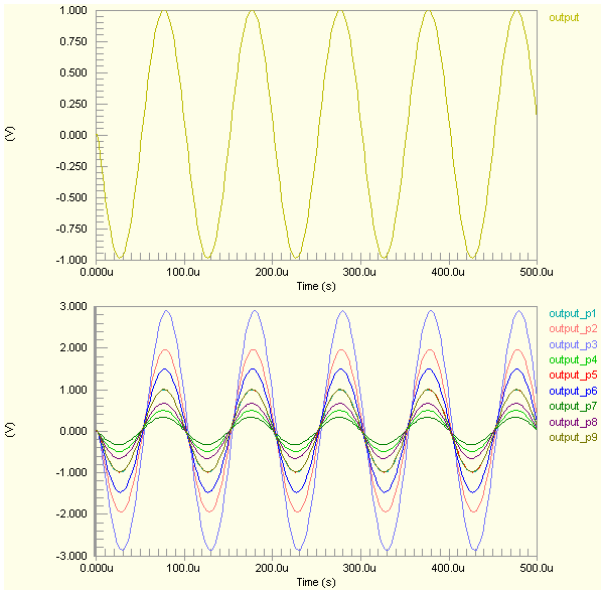
- output_p4
- output_p5
- output_p6
- output_p7
- output_p8
- output_p9

The default value waveform (out) will also be generated for comparison. Hence, running the simulation will yield the output waveforms shown in the following images.

AC Small Signal analysis



Transient analysis



Temperature Sweep

Description

The Temperature Sweep feature is used to analyze the circuit at each temperature in a specified range, producing a series a curves, one for each temperature setting. The Simulator performs multiple passes of any of the standard analyses that are enabled (AC, DC Sweep, Operating Point, Transient, Transfer Function, Noise).

Setup

Temperature Sweep is set up on the **Temperature Sweep Setup** page of the *Analyses Setup* dialog (after the dialog appears, simply click the **Temperature Sweep** entry in the **Analyses/Options** list). An example setup for this feature is shown in the image below:

Temperature Sweep Setup	
Parameter	Value
Start Temparature	0.000
Stop Temparature	100.0
Step Temperature	25.00

Parameters

- **Start Temperature** - the initial temperature of the required sweep range (in Degrees C).
- **Stop Temperature** - the final temperature of the required sweep range (in Degrees C).
- **Step Temperature** - the incremental step to be used in determining the sweep values across the defined sweep range.

Notes

At least one of the standard analysis types (AC, DC Sweep, Operating Point, Transient, Transfer Function, Noise) must be enabled in order to perform a Temperature Sweep analysis.

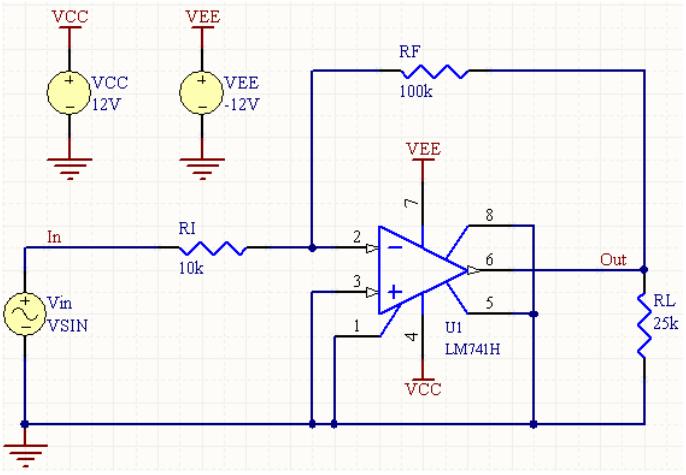
Data is saved for all signals in the **Available Signals** list, on the **General Setup** page of the *Analyses Setup* dialog.

Running a Temperature Sweep can result in a large amount of data being calculated. To limit the amount of data calculated, you can set the **Collect Data For** option on the **General Setup** page of the *Analyses Setup* dialog to **Active Signals**. With this option, data is only calculated for variables currently listed in the **Active Signals** list.

Temperature can also be varied using a Parameter Sweep. This is useful if you want to vary the temperature as either the primary or secondary parameter in a two-parameter sweep.

As running a Temperature Sweep actually performs multiple passes of the analysis (using a different value for the temperature with each pass), there is a special identifier used when displaying the waveforms in the Sim Data Editor's Waveform Analysis window. Each pass is identified by adding a letter and number as a suffix to the waveform name. For a Temperature Sweep, the letter used is `t` and the number used identifies which pass the waveform relates to (e.g. `Output_t1`, `Output_t2`, etc).

Examples



Consider the circuit in the image above, where an AC Small Signal analysis is to be performed in conjunction with the use of the Temperature Sweep feature. The AC Small Signal analysis is defined with the following parameters:

- **Start Frequency** = `1.000`
- **Stop Frequency** = `1.000g`
- **Sweep Type** = `Decade`
- **Test Points** = `100`
- **Total Test Points** = `901`.

The Temperature Sweep is defined with the following parameter values:

- **Start Temperature** = `0.000`

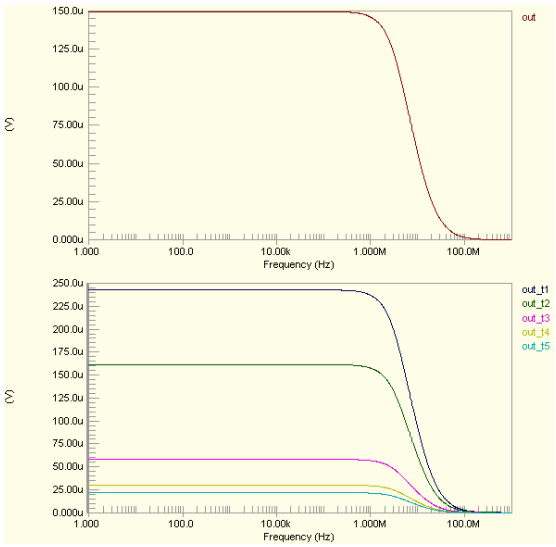
- **Stop Temperature** = `100.0`
- **Step Temperature** = `25.00`.

The entry in the SPICE netlist will be:

```
*Selected Circuit Analyses:
.AC DEC 100 1 1E9
.CONTROL
SWEEP OPTION[TEMP] 0 100 25
.ENDC
```

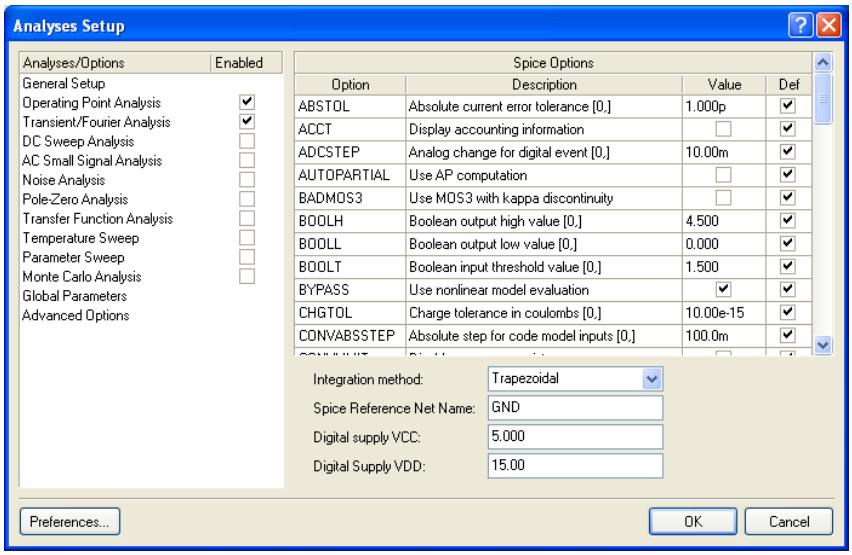
There will be five waveforms in all generated by the sweep (five different values for temperature across the defined sweep range, resulting in five separate simulation passes). The default value waveform will also be generated for comparison. Hence, running the simulation will yield the following waveforms with respect to the Out node:

- `out`
- `out_t1`
- `out_t2`
- `out_t3`
- `out_t4`
- `out_t5`.



Advanced SPICE Options

The **SPICE Options** page of the *Analyses Setup* dialog enables you to define advanced simulation options, including the values of SPICE variables, the integration method used by the Simulation Engine and the simulation reference net.



You can define the numerical integration method used for simulations in the **Integration method** field. The `Trapezoidal` method is relatively fast and accurate, but tends to oscillate under certain conditions. The `Gear` methods require longer simulation times, but tend to be more stable. Using a higher gear order theoretically leads to more accurate results, but increases simulation time. The default method is `Trapezoidal`.

All of the digital components supplied in the component libraries have hidden power pins (VCC and GND for the TTL devices, and VDD and GND for the CMOS series devices). These hidden power pins are automatically connected during netlisting and assigned the voltages specified in the **Digital Supply VCC** and **Digital Supply VDD** fields. To change the default power supply values, enter new values in these fields. The defaults are `VCC = 5`, `VDD = 15`.

To power any digital components in your circuit from nets other than VCC (or VDD for CMOS) you must include source components to create the appropriate voltages, un-hide the power pins for each component and wire the power pins to the appropriate power nets.

When a simulation is run, all data that is collected for all available signals is referenced to a specific net in the circuit. This net is defined in the **Spice Reference Net Name** field and, by default, is the `GND` net. To run a transient simulation which references a net other than ground, enter the net name in this field.

The main area of this page of the dialog lists options that give you direct access to SPICE variables, from where you can change iteration limits, error tolerances, etc.

To change the value of a SPICE variable, click inside the associated **Value** column entry, edit the value as required, then press **Enter** or click outside of the column entry, to change the variable to the new value.

To return an option to its default value, simply enable the associated entry in the **Def** column.

The following details each of the SPICE variables contained in the list:

SPICE Option	Description	Default Value
ABSTOL	Sets the absolute current error tolerance (in Amps).	1.000p
ACCT	Causes accounting and run-time statistics to be displayed.	Disabled
ADCSTEP	Minimum step size required to register an event on the input of the internal A/D converters.	10.00m
AUTOPARTIAL	Enables automatic computation of partial derivatives for XSpice code modules.	Disabled
BADMOS3	Uses the older version of the MOS3 model with the "kappa" discontinuity.	Disabled
BOOLH	Sets the high output level of a Boolean expression.	4.500
BOOLL	Sets the low output level of a Boolean expression.	0.000
BOOLT	Sets the input threshold level of a Boolean expression.	1.500

SPICE Option	Description	Default Value
BYPASS	Enables device bypass scheme for nonlinear model evaluation.	Enabled
CHGTOL	Provides lower limit on capacitor charge or inductor flux (in Coulombs); used in the LTE timestep control algorithm.	10.00e-15
CONVABSSTEP	Sets limit of the absolute step size in solving for the DC operating point convergence for code model inputs.	100.0m
CONVLIMIT	Disables convergence algorithm used in some built-in component models.	Disabled
CONVSTEP	Sets the limit of the relative step size in solving for the DC operating point convergence for code model inputs.	250.0m
CURRENTMNS	Sets scale factor used to determine min supply current when value not specified in SimCode model.	1.500
CURRENTMXS	Scale factor used to determine max supply current when value not specified in SimCode model.	500.0m
DEFAD	Sets the MOS drain diffusion area.	0.000
DEFAS	Sets the MOS source diffusion area.	0.000
DEFL	Sets the MOS channel length (in micrometers).	100.0
DEFW	Sets the MOS channel width (in micrometers).	100.0
DRIVEMNS	Sets scale factor used to determine min output drive capacity when value not specified in SimCode model.	1.500
DRIVEMXS	Sets scale factor used to determine max output drive capacity when value is not specified in SimCode model.	500.0m
DRVMNTYMX	Temporary global override for output drive capacity index on SimCode devices (None, Minimum, Typical, Maximum).	None
GMIN	Sets min conductance (max resistance) of any device in the circuit (in mhos). Also sets value of the conductance that is placed in parallel with each pn junction in the circuit.	1.000p
GMINSTEP	Sets the number of steps in the GMIN stepping algorithm. When set to 0, GMIN stepping is disabled, making source stepping the simulator's default DC (operating point) convergence algorithm.	10
IMNTYMX	Temporary global override for supply current index on SimCode devices (None, Minimum, Typical, Maximum).	None
ITL1	Sets Operating Point Analysis iteration limit.	100
ITL2	Sets DC Analysis iteration limit.	50
ITL3	Sets lower Transient Analysis iteration limit.	4
ITL4	Sets Transient Analysis timepoint iteration limit.	40
ITL5	Sets Transient Analysis total iteration limit.	5000
KEEPOPINFO	Retains operating point information when an AC Analysis is run.	Disabled
LDMNTYMX	Temporary global override for input loading index on SimCode devices (None, Minimum, Typical, Maximum).	None
LIST	Displays a comprehensive list of all elements in the circuit with	Disabled

SPICE Option	Description	Default Value
	connectivity and values.	
LOADMNS	Sets scale factor used to determine min input loading (max input resistance) when value not specified in SimCode model.	1.500
LOADMXS	Sets scale factor used to determine max input loading (min input resistance) when value not specified in SimCode model.	500.0m
MAXEVTITER	Sets the max number of event iterations for DC (operating point) convergence.	0
MAXOPALTER	Sets the max number of analog/event alternations for DC (operating point) convergence.	0
MINBREAK	Sets the min time between breakpoints (in seconds).	0 (Automatic)
NOOPALTER	Enables DC (operating point) alternations.	Disabled
NOOPITER	Skip directly to GMIN stepping algorithm.	Disabled
OPTS	Displays a list of all standard SPICE3 Option parameter settings.	Disabled
PIVREL	Sets relative ratio between the largest column entry in the matrix and an acceptable pivot value. The value must be between 0 and 1.	1.000m
PIVTOL	Sets the absolute min value for a matrix entry to be accepted as a pivot.	100.0e-15
PROPMNS	Sets scale factor used to determine min propagation delay when value is not specified in SimCode model.	500.0m
PROPMXS	Sets scale factor used to determine max propagation delay when value is not specified in SimCode model.	1.500
RAMPTIME	Controls turn-on time of independent sources and capacitor and inductor initial conditions from zero to their final value during the time period specified (in seconds).	0.000
RELTOL	Sets relative error tolerance of the program. The value must be between 0 and 1.	1.000m
RSHUNT	Value (in ohms) of resistors added between each circuit node and ground, helping to eliminate problems such as "singular matrix" errors. In general, the value of RSHUNT should be set to a very high resistance (1e+12).	0.000 (No shunt resistors)
SIMWARN	Allows SimCode warning messages to be displayed at run time. SimCode warnings may include information concerning timing violations (tsetup, thold, etc.) or indicate supply voltage dropping below device specifications (None, No, Yes).	None
SRCSTEP	Sets the number of steps in the source stepping algorithm for DC (operating point) convergence.	10
TEMP	Sets the actual operating temperature of the circuit (in Degrees C). Any deviation from TNOM will produce a change in the simulation results. Where a device model has a `Temperature` parameter that can be set at the component-level, setting a value for that parameter will override TEMP.	27.00
TNOM	Sets the nominal temperature for which device models are created (in Degrees C). Where a device model has a `TNOM` parameter that can be set at the model file-level, setting a value for that parameter will	27.00

SPICE Option	Description	Default Value
	override TNOM.	
TPMNTYMX	Temporary global override for propagation delay index on SimCode devices (None, Minimum, Typical, Maximum).	None
TRTOL	Used in the LTE timestep control algorithm. This is an estimate of the factor by which SPICE overestimates the actual truncation error.	7.000
TRYTOCOMPACT	Applicable to the LTRA model. When specified, the simulator tries to condense LTRA transmission line's past history of input voltages and currents	Disabled
VNTOL	Sets the absolute voltage tolerance of the program (in Volts).	1.000u

Notes

In general, you should not have to change any of the advanced SPICE parameters in this page of the dialog for accurate simulation. Only change these options if you have a good understanding of SPICE simulation parameters.

When troubleshooting Transient analysis failure, try setting:

- `ABSTOL=RELTOL * (lowest current magnitude in the circuit)`
- `VNTOL= RELTOL *  (lowest voltage magnitude in the circuit)`

Raising the value of `GMIN` may help with convergence, but decreases accuracy.

`ITL1` may need to be raised as high as `500` for many circuits.

`ITL2` may need to be raised as high as `200` for some circuits.

`ITL3` is not implemented in SPICE3. It is provided for compatibility in creating SPICE2 netlists.

Raising `ITL4` to `100` or more may help to eliminate "timestep too small" errors improving both convergence and speed.

`ITL5` is not implemented in SPICE3. It is provided for compatibility in creating SPICE2 netlists.

Enabling the `KEEPOPINFO` option is useful if the circuit is large and you do not want to run a redundant Operating Point Analysis.

In the numerical pivoting algorithm, the allowed min pivot is determined by:

```
EPSREL=AMAX1(PIVREL * MAXVAL, PIVTOL)
```

where `MAXVAL` is the max element in the column where a pivot is sought (partial pivoting).

With respect to the `RELTOL` option, larger values mean faster simulation time, but less accuracy.

Simulation Troubleshooting

When a circuit will not simulate you must identify if the problem is in the circuit, or the process of simulation. Follow the information contained in this section of the reference and work through the suggested points, trying one at a time.

Sometimes during a simulation a message will be displayed reporting errors or warnings. These messages are listed in the **Messages** panel.

Warning Messages

Warning messages are not fatal to the simulation. They generally provide information about changes that SPICE had to make to the circuit in order to complete the simulation. These include invalid or missing parameters, and so on.

Digital SimCode warnings may include information such as timing violations (tsetup, thold, trec, tw, etc.) or significant drops in power supply voltage on digital components.

Valid simulation results are normally generated even if warnings are reported.

Error Messages

Error messages provide information about problems that SPICE could not resolve and were fatal to the simulation. Error messages indicate that simulation results could not be generated, so they must be corrected before you will be able to analyze the circuit.

Troubleshooting netlist generation failure

When you run a simulation, the first thing that happens is the circuit is analyzed and a SPICE netlist is generated. This netlist is then passed to the SPICE engine, which simulates the circuit and generates the results.

Any errors that are detected during netlisting are listed in the **Messages** panel. Likely causes of netlisting errors include:

- A component in the schematic source document(s) not containing simulation information. To check if a component is suitable for simulation, double-click on the component in the schematic to open its *Component Properties* dialog and confirm that there is a linked simulation model, in the **Models** region of the dialog.
- The simulation model file that a component references is not in the location specified iin the **Model Location** region on the **Model Kind** tab of the *Sim Model* dialog. This could happen if the associated integrated library in which the model is stored is not installed, or it has been moved from its original install location.

  All source component libraries are installed to the following location:

  ```
  \Library
  ```

  This root folder includes various sub-folders containing component integrated libraries from specific manufacturers, as well as two general integrated library files (`Miscellaneous Devices.IntLib` and `Miscellaneous Connectors.IntLib`) providing general schematic components (many of which are simulation-ready).

  The `\Library\Sim` folder contains various `.txt` and `.scb` files, for SimCode-based simulation models, such as CMOS and 74XX series digital component models.

  The `\Library\Simulation` folder contains the following specific simulation-ready component integrated libraries:

  ```
  Simulation Math Function.IntLib
  Simulation Sources.IntLib
  Simulation Special Function.IntLib
  Simulation Transmission Line.IntLib
  ```

- The path to the Digital SimCode model (uncompiled source file (`*.txt`) or compiled file (`*.scb`)), referred to as `{MODEL_PATH}`, does not match the location of the model. This could happen if the model is moved to a different location on the hard drive. The model path is defined in the *Simulation Preferences* dialog and is relative to the `Library` folder of the installation. By default, the path is `\Library\Sim\`.

Trouble-shooting simulation analysis failures

One of the challenges of all Simulators is convergence. What exactly is meant by the term, “convergence”? Like most Simulators, the Altium Designer-based Simulator's SPICE engine uses an iterative process of repeatedly solving the equations that represent your circuit, to find the quiescent circuit voltages and currents. If it fails to find these voltages and current (fails to converge) then it will not be able to perform an analysis of the circuit.

SPICE uses simultaneous linear equations, expressed in matrix form, to determine the operating point (DC voltages and currents) of a circuit at each step of the simulation. The circuit is reduced to an array of conductances which are placed in the

matrix to form the equations (`G * V = I`). When a circuit includes nonlinear elements, SPICE uses multiple iterations of the linear equations to account for the non-linearities. SPICE makes an initial guess at the node voltages then calculates the branch currents based on the conductances in the circuit. SPICE then uses the branch currents to recalculate the node voltages, and the cycle is repeated. This cycle continues until all of the node voltages and branch currents fall within specified tolerances (converge).

However, if the voltages or currents do not converge within a specified number of iterations, SPICE produces error messages (such as "`singular matrix`", "`Gmin stepping failed`", "`source stepping failed`" or "`iteration limit reached`") and aborts the simulation. SPICE uses the results of each simulation step as the initial guesses for the next step. If you are performing a Transient analysis (that is, time is being stepped) and SPICE cannot converge on a solution using the specified timestep, the timestep is automatically reduced, and the cycle is repeated. If the timestep is reduced too far, SPICE displays a "`Timestep too small`" message and aborts the simulation.

General simulation convergence troubleshooting

When a simulation analysis fails, the most common problem is failure of the circuit to converge to a sensible operating point. Use the following techniques to solve convergence problems.

Convergence trouble-shooting steps

- When you have a convergence problem, first turn off all the analyses except the Operating Point analysis.
- Consult the **Messages** panel for any errors/warnings relating to simulation.
- Make sure the circuit is wired correctly. Dangling nodes and stray parts are not allowed.
- Ensure that the circuit has a ground node, and that every node in the circuit has a DC path to this ground. Components that can isolate a node include transformers and capacitors. Voltage sources are considered a DC short circuit, current sources are considered a DC open circuit.
- Ensure that zeros have not been confused with the letter O when entering simulation parameters.
- Ensure that proper SPICE multipliers have been specified (MEG instead of M for 1E+6) for any component values or simulation parameters. Multipliers are not case sensitive. Also, spaces between values and multipliers are not allowed. For example it should be 1.0uF, not 1.0 uF.
- Make sure all devices and sources are set to their proper values.
- Make sure the gain of any dependent source is correctly set.
- Temporarily eliminate series capacitors or current sources and re-run the simulation.
- Temporarily eliminate parallel inductors or voltage sources and re-run the simulation.
- On the **SPICE Options** page of the *Analyses Setup* dialog (from the schematic select **Design » Simulate » Mixed Sim**, then click the **Advanced Options** entry in the **Analyses/Options** list), increase the value of the `ITL1` parameter to `300`. This will allow the Operating Point analysis to go through more iterations before giving up.
- Add .NS (Nodeset) devices to define the node voltages. If the initial guess of a node voltage is way off, the .NS device can be used to predefine a starting voltage that is used for a preliminary pass of the operating point analysis.
- If the Nodeset device does not assist in convergence, try defining the initial conditions by placing .IC devices. In this case the node voltages are held at the specified values during the Operating Point analysis, then released during the Transient analysis.
- Enable the **Use Initial Conditions** option on the **Transient/Fourier Analysis Setup** page of the *Analyses Setup* dialog (from the schematic, select **Design » Simulate » Mixed Sim**, then click the **Transient/Fourier Analysis** entry in the **Analyses/Options** list). This option works in conjunction with the .IC devices (or the IC parameter of the components). By setting this option, the Operating Point analysis is not performed and the specified voltages are used as the initial conditions for the Transient analysis.
- Specify the series resistance parameters of your models and increase the `GMIN` option (**SPICE Options** page of the *Analyses Setup* dialog) by a factor of `10`. Specify the initial condition of semiconductor devices, especially diodes, as `OFF`.

DC Sweep Analysis troubleshooting

When you have a problem with a DC Sweep analysis, first try the steps listed in the *General simulation convergence troubleshooting* section.

If you still encounter problems, try the following:

- Change the value of the `Primary Step` parameter on the **DC Sweep Analysis** page of the *Analyses Setup* dialog. If discontinuities exist in a device model (perhaps between the linear and saturation regions of the model), increasing the step

size may allow the simulation to step over the discontinuity. Making the steps smaller, on the other hand, will allow the simulation to resolve rapid voltage-transition discontinuities.

- Disable the DC Sweep analysis. Some problems (such as hysteresis) cannot be resolved by DC analysis. In such cases, it is more effective to use the Transient analysis and ramp the values of the appropriate power sources.

Transient Analysis troubleshooting

When you have a problem with a Transient analysis, first try the steps listed in the *General simulation convergence troubleshooting* section.

If you still encounter problems, try the following.

On the **SPICE Options** page of the *Analyses Setup* dialog (from the schematic select **Design » Simulate » Mixed Sim**, then click the **Advanced Options** entry in the **Analyses/Options** list):

- Set the `RELTOL` parameter to `0.01`. By increasing the tolerance from its default of `0.001` (0.1% accuracy), fewer iterations will be required to converge on a solution and the simulation will complete much more quickly.
- Increase the value of the `ITL4` parameter to `100`. This will allow the Transient analysis to go through more iterations for each timestep before giving up. Raising this value may help to eliminate “timestep too small” errors improving both convergence and simulation speed.
- Reduce the accuracy by increasing the values of `ABSTOL` and `VNTOL`, if current/voltage levels allow. Your particular circuit may not require resolutions down to 1uV or 1pA. You should, however, allow at least an order of magnitude below the lowest expected voltage or current levels of your circuit.
- change the **Integration Method** to one of the `Gear` methods. Gear integration requires a longer simulation time, but is generally more stable than trapezoidal. Gear integration may be particularly useful with circuits that oscillate or have feedback paths.

Additional things to try:

- Realistically model your circuit. Add realistic parasitics, especially stray/junction capacitance. Use RC snubbers around diodes. Replace device models with subcircuits, especially for RF and power devices.
- Increase the rise/fall times of any Periodic Pulse sources in your circuit. Even the best pulse generators cannot switch instantaneously.

Revision History

Date	Version No.	Revision
01-Dec-2004	1.0	New product release
20-May-2005	1.1	Updated for SP4
12-Dec-2005	1.2	Path references updated for Altium Designer 6
17-Jan-2006	1.3	Updated with PSpice support information
09-Jun-2006	1.4	Updated for Altium Designer 6.3. Added definitions table for Available Signals that appear on the General Setup page of the Analyses Setup dialog.
28-Aug-2006	1.5	Added parameter information for BSIM3 MOSFET model support. Added parameter information for EKV MOSFET model support.
21-Apr-2008	1.6	Updated Page Size to A4.

Software, hardware, documentation and related materials:

Copyright © 2008 Altium Limited.

All rights reserved. You are permitted to print this document provided that (1) the use of such is for personal use only and will not be copied or posted on any network computer or broadcast in any media and (2) no modifications of the document is made. Unauthorized duplication, in whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Altium Limited. Unauthorized duplication of this work may also be prohibited by local statute. Violators may be subject to both criminal and civil penalties, including fines and/or imprisonment. Altium, Altium Designer, Board Insight, CAMtastic, CircuitStudio, Design Explorer, DXP, LiveDesign, NanoBoard, NanoTalk, Nexar, nVisage, P-CAD, Protel, SimCode, Situs, TASKING and Topological Autorouting and their respective logos are trademarks or registered trademarks of Altium Limited or its subsidiaries. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.